DOCUMENT RESUME

ED 146 397 CE 013 351

AUTHOR	Adams, George C.
TITLE	Dimensioning, Tolerancing, and Machine Finishes.
INSTITUTION	Rutgers, The State Univ., New Brunswick, N.J. Curriculum Lab.
SPONS AGENCY	New Jersey State Dept. of Education, Trenton. Div. of Vocational Education.
PUB DATE	Nov 77
NOTE	138p.; For a related document see ED 120 550
AVAILABLE FROM	New Jersey Vocational-Technical Curriculum Laboratory, Building 4103, Kilmer Campus, Rutgers University, New Brunswick, New Jersey 08903 ($3.75 plus postage)
EDRS PRICE	MF-$0.83 HC-$7.35 Plus Postage.
DESCRIPTORS	*Drafting; *Engineering Drawing; Engineering Graphics; Geometric Concepts; Guides; High School Curriculum; Industry; *Machine Tools; *Measurement; Secondary Education; Study Guides; *Technical Illustration; Vocational Education

ABSTRACT

Intended for use with the vocational education student interested in technical drawing, this guide provides answers to questions relating to dimensioning and tolerancing machine drawings. It also gives examples of standard dimensioning practices, tolerancing applications, and finish applications. The problems and examples presented are based on the most commonly accepted practices in industry and their applications. Content is divided into ten units, titled as follows: Measurement; Dimensioning--Size, Shape, Location; Dimensioning Systems; Production Processes; Sequence of Operations; Machine Finishes; Finish Marks; Tolerance; and Tolerance Applications. Each unit includes from one to nine sections, illustrations, and review questions. (SH)

****
* Documents acquired by ERIC include many informal unpublished *
* materials not available from other sources. ERIC makes every effort *
* to obtain the best copy available. Nevertheless, items of marginal *
* reproducibility are often encountered and this affects the quality *
* of the microfiche and hardcopy reproductions ERIC makes available *
* via the ERIC Document Reproduction Service (EDRS). EDRS is not *
* responsible for the quality of the original document. Reproductions *
* supplied by EDRS are the best that can be made from the original. *
****

ED146397

State of New Jersey
Department of Education
Division of Vocational Education

DIMENSIONING, TOLERANCING, AND MACHINE FINISHES

U S DEPARTMENT OF HEALTH
EDUCATION & WELFARE
NATIONAL INSTITUTE OF
EDUCATION

THIS DOCUMENT HAS BEEN REPRODUCED EXACTLY AS RECEIVED FROM THE PERSON OR ORGANIZATION ORIGINATING IT. POINTS OF VIEW OR OPINIONS STATED DO NOT NECESSARILY REPRESENT OFFICIAL NATIONAL INSTITUTE OF EDUCATION POSITION OR POLICY

Prepared by:
George C. Adams, Instructor

Joseph W. English, Director
Gloucester County Vocational-Technical School
Sewell, New Jersey

"PERMISSION TO REPRODUCE THIS MATERIAL HAS BEEN GRANTED BY

Philip L. Edgecomb

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC) AND USERS OF THE ERIC SYSTEM."

Vocational-Technical
Curriculum Laboratory
Rutgers – The State University
Building 4103 – Kilmer Campus
New Brunswick, New Jersey

CE 013 351

November 1977

NEW JERSEY DEPARTMENT OF EDUCATION

FRED G. BURKE, COMMISSIONER

RALPH H. LATAILLE, DEPUTY COMMISSIONER

DIVISION OF VOCATIONAL EDUCATION

WILLIAM WENZEL, ASSISTANT COMMISSIONER

JOHN E. RADVANY, DEPUTY ASSISTANT COMMISSIONER

HAROLD R. SELTZER, DEPUTY ASSISTANT COMMISSIONER (Acting)

JOSEPH F. KELLY, DIRECTOR

BUREAU OF OCCUPATIONAL AND CAREER RESEARCH DEVELOPMENT

FREDERICK L. CAPPELLO, DIRECTOR

VOCATIONAL CURRICULUM AND MEDIA DEVELOPMENT

RUTGERS – THE STATE UNIVERSITY

PHILIP L. EDGECOMB, DIRECTOR (Acting)

NEW JERSEY VOCATIONAL-TECHNICAL

CURRICULUM LABORATORY

RUTGERS – THE STATE UNIVERSITY

BUILDING 4103 – KILMER CAMPUS

NEW BRUNSWICK, NEW JERSEY

INTRODUCTION

The purpose of this workbook is to answer some of the questions you are going to run into while dimensioning and tolerancing machine drawings. It will be a source of standard information, definitions, and other resources. It will also give examples of standard dimensioning practices, tolerancing applications, and finish applications. You will have to use your own judgement when putting these examples into practice on your drawings.

It would be impossible to incorporate examples of all the standards used in industry, since these vary between companies and areas of specialization. The problems encountered are equally unique and must be solved individually, based on the standards of the individual company or industry. The solutions should be logically ordered, yielding the most economical results —— both in time and money. The problems and examples presented to you will be based on the most commonly accepted practices of industry and their applications. These can be expanded later to meet the needs of a specific industry.

It is suggested that this book may be used concurrently with *Introduction to Geometric Dimensioning and Tolerancing* by Donald C. Proebstle, published by the New Jersey Vocational–Technical Curriculum Laboratory.

TABLE OF CONTENTS

Page

Introduction i

Unit I: Measurement 1

Unit II: Dimensioning 7

- Extension Lines 8
- Dimension Lines 9
- Leader Lines 10
- Usage 10
- Local Notes 13
- General Notes 14
- Metric vs. English System 15
- Dual Dimensioning 19
- Angles 21

Unit III: Dimensioning – Size, Shape, Location 26

- Size 26
- Shape 28
- Location 30
- Rectangular Coordinates 30
- Polar or Angular Dimensioning 31
- Chordal Dimensioning 32

Unit IV: Dimensioning Systems 36

- Unidirectional System 37
- Aligned System 38
- Chain Dimensions 40
- Baseline (Datum-Line) Dimensioning 41
- Numerical Control Dimensioning 44
- True-Position Dimensioning 47

Unit V: Production Processes 55

- Casting 56
- Rolling 59
- Forging 60
- Machining 62
- Drilling 63
- Reaming 65
- Broaching 67

Boring 68
Counterboring, Countersinking 70
Spotfacing 73
Lathe Work 74
Straight Turning 75
Taper Turning 76
Chamfers 78
Knurling 79
Threads 80
Milling 81
Planing and Shaping 83

Unit VI: Sequence of Operations 88

Making a Hoze Nozzle 88

Unit VII: Machine Finishes 100

Terms Used in Machine Finishes 101

Unit VIII: Finish Marks 103

Different Types Used 103
Roughness Height 104
Waviness Height 105
Waviness Width 106
Lay Symbols 107
Roughness-Width Cut-Off 107
Contact or Bearing Area 108

Unit IX: Tolerance 109

Terms Used 109
Methods of Indicating 110

Unit X: Tolerance Applications 112

Standard Tolerances 113
Plus–Minus Method 116
Limits 117
Fits 118
Clearance Fit 118
Transition Fit 119
Interference Fit 120
Running and Sliding Fits (RC) 121

Locational Clearance (LC) 122
Locational Transition (LT) 122
Locational Interference (LN) 123
Force or Shrink (FN) 123
Applications 124

Appendix

Abbreviations Used on Drawings 127
Abbreviations Relating to Screw Threads 131
Engineering Societies, Trade Associations, Inspection Bureaus, etc. 131

UNIT I: MEASUREMENT

WHAT IS MEASUREMENT?

Measurement is a derived quantity which answers the question "How Much" or "How Long". These measurements or answers are based on agreements which have been accepted over a period of time as standards.

A day is accepted as one revolution of the earth on its axis, or sunrise to sunrise. An hour is 1/24th of a day. These measurements are accepted as standard, even though they are based on natural occurrences. An hour could have been set as 1/10th or 1/20th of a day, so they are derived or obtained from some source -- in this case the natural turning of the earth.

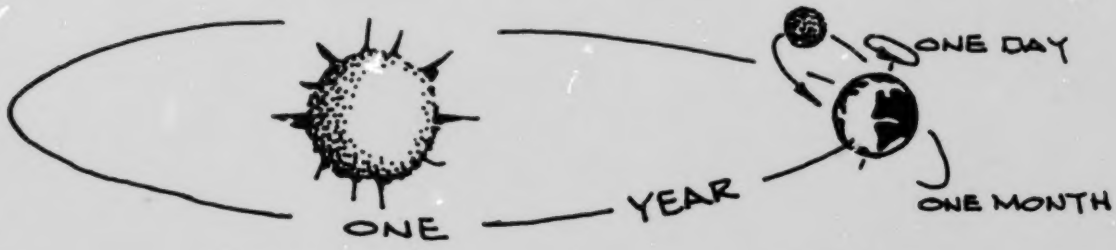


FIG. 1-1

WHERE DO MEASUREMENTS COME FROM?

As was mentioned before, measurements are obtained from some accepted source. This source originally occurred naturally. When man started trading between groups, systems of barter were established. Twelve coconuts might equal one bag of corn. A cowmight equal three sheep.

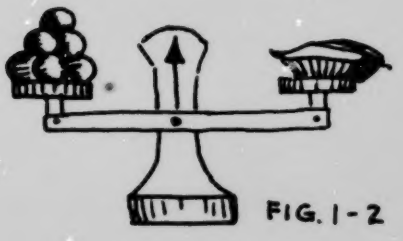


FIG. 1-2

After a while, this type of trading could become quite complex, especially if you had to trade with a number of people.

To help in trade, standard weights were set up. Most of these weights were based on natural items such as stones, bags of grain, or pieces of precious metals.

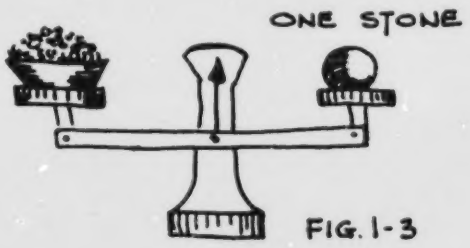


FIG. 1-3

Over time they became standardized and more rigidly defined. Weights became more rigidly defined, because it would be easy to use "long weights" (heavier) when buying and "short weights" (lighter) when selling.

The standards were usually established and enforced by the ruling body of an area. Agreements between areas were established by treaty, and trade began.

FIG. 1-4

WHAT IMPORTANCE DOES THIS HAVE TODAY?

The standard measurements used in America today were brought from England and Europe.

FIG. 1-5

The cubit was used during biblical times and was equal to the length of a man's forearm, or approximately 18" (1/2 yard). The Egyptians standardized the cubit and made a "royal cubit" of black granite for a permanent record.

The yard was equal to the distance from the King's nose to the end of his fingers, or approximately 36". This was recorded on bronze and established as a standard by Queen Elizabeth in 1558.

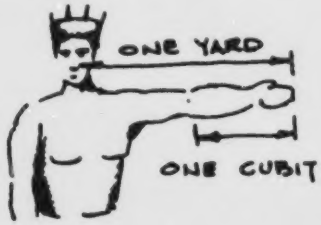
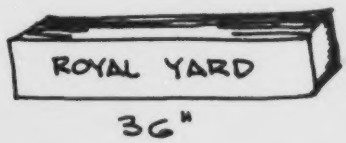


FIG. 1-6



The foot was equal to the King's foot; and the inch was equal to "three barley corns, round and dry", or the width of the King's thumb.

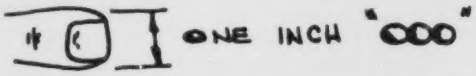


FIG. 1-7

You should be able to notice some of the problems with these standards. Each new king brought a new set of measurements. If the king was 6'6" you would get more than if the king was 4'6". It was because of these problems that measurements were put down on stable materials and used as a check against other measurements.

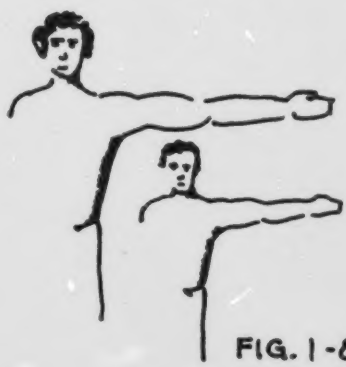


FIG. 1-8

In 1824 the English established the present day yard as their standard measurement, and in 1793, under Napoleon's reign, the meter was established as the standard measurement in France. Both of these are still used, although the meter or Metric system is becoming the more widely accepted of the two.

WHY CONVERT TO METRICS?

The United States has been planning to convert to the Metric system for some time and has set the 1980's as the deadline for the conversion. The major reason for the changeover is economic -- money. People have accepted the Metric system throughout the world as an easier one to use and teach than the English system. United States'

products sold to those countries had to be made either to Metric dimensions, or English-dimensioned tools had to be sold with the equipment. Both cost more than if it were made to Metric sizes for both countries. Also, during World War II the need for international standards became evident. United States and British equipment lay idle because simple parts like nuts and bolts couldn't even be interchanged. Lives and battles were lost because the dimensioning systems were not the same.

FIG. 1-9

To fill this need, the International Standards Organization (ISO) was formed and worked closely with the American National Standards Institute (ANSI) and other groups to incorporate existing standards into a world-wide agreement. One of those agreements was a standardized dimensioning system: The Metric system.

INTERNATIONAL
STANDARDS
ORGANIZATION

FIG. 1-10

THINK ABOUT IT!

Look at the other measurements we use and check their origins.

Besides the normal measurements, can you trace the origins of slang expressions like "a stone's throw"?

What are you doing to make the conversion from the English to Metric system? Can you give examples of changes which have already occurred?

Can you name organizations other than those mentioned which standardize sizes and dimensions?

EXERCISE YOUR BRAIN

What is a measurement?

Why is it necessary to have standard measurements?

How did the present-day measurements come about?

Who assumed the responsibility for setting standards and enforcing them?

What were the reasons for establishing international standards of measurement?

What organization is responsible for setting the international standards? What American organization serves the same function within the United States?

UNIT II: DIMENSIONING

WHAT'S HE SAYING?

Before you can talk "dimensioning", you'd better make sure you can speak the language. If you've ever been someplace where everyone else is speaking a language you can't understand, you know what a lonely and frustrating feeling it is. Each technical field has its own language and slang. Without the ability to communicate, you are lost. If you don't want to take the time to learn the language, don't complain when you end up in left field instead of right field.

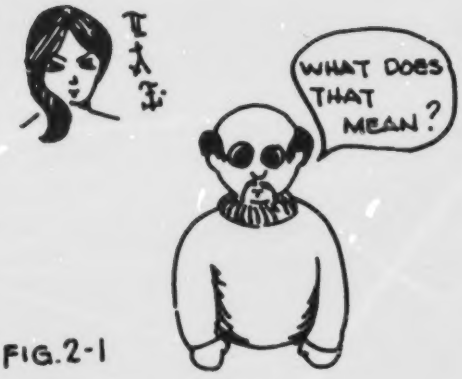


FIG.2-1

Basically there are three types of terms you need to know:

1. Graphical – These are the symbols, lines, and standard practices used on your drawings.

2. Technical – These are the "book meanings" you will be using in conversations reports, and tests. These are the hardest to understand and use correctly.

3. Notational – These are the general and special notes you will use on drawings, reports, and specifications. You will put into practice your understanding of "book meanings", abbreviations, and standard practices when making notations.

The specific words or terms you will be using in this workbook will be explained as they come up. You can also find them in the glossary in the back of the workbook or in your textbook. Take the time to know what you're talking about.

WHAT'S AN ABBR.?

An abbreviation (abbr.) is a shortening of a word. These shortened words can be a great help and time saver, or the cause of a major disaster. *Don't use just any abbreviation!*

Abbreviations, like many other technical items, have been standardized -- *use those standards!* If you are in doubt as to the standard abbreviation of a word, look it up in a dictionary, your textbook, or the back of this workbook. If you can't find the standard abbreviation, or still aren't sure -- spell out the word: don't make one up.

FIG. 2-2

WHAT'S IT GOING TO BE?

Each drawing you make serves a purpose -- to tell the craftsman *what* he is going to make, out of *what* material, using *what* method. You are telling the story of that part and its function, just as a writer tells a story. In telling the story of a machine part, you are using symbols, dimensions, and notes.

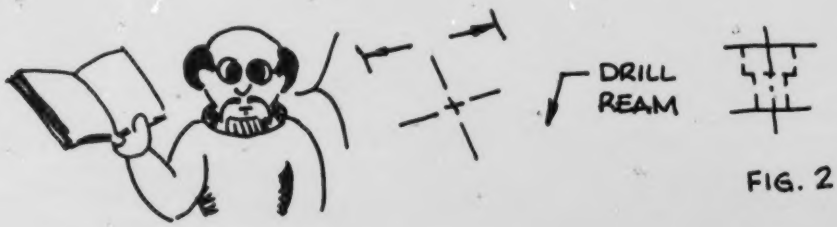


FIG. 2-3

The symbols you would use in describing a machine part are the ABC's -- the alphabet of lines. Besides the lines you use to show the outline and hidden surfaces of the object, you will be using extension, dimension, and leader lines to fully describe the *size* and *shape* of the part to the craftsman.

Extension lines are thin, unbroken lines that extend from the object or feature to be dimensioned and help to show what is being dimensioned, and how it is being dimensioned.

FIG. 2-4

Extension lines start 1/32" to 1/16" from the object or feature and extend unbroken 1/8" beyond the dimension line. Center lines (℄) may be used as extension lines. When a center line is used as an extension line, you don't break it when you cross an object line. If a center line is extended any distance, it should still look like a center line to prevent confusion.

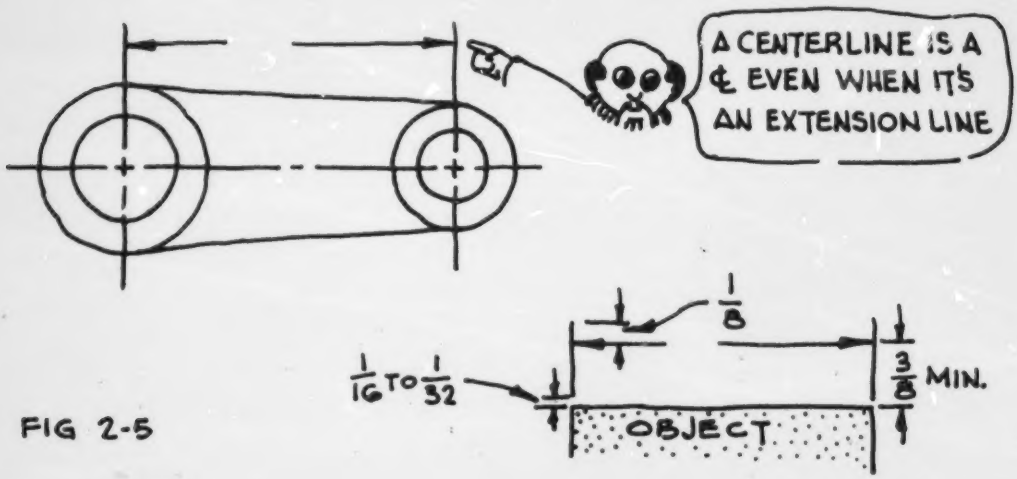
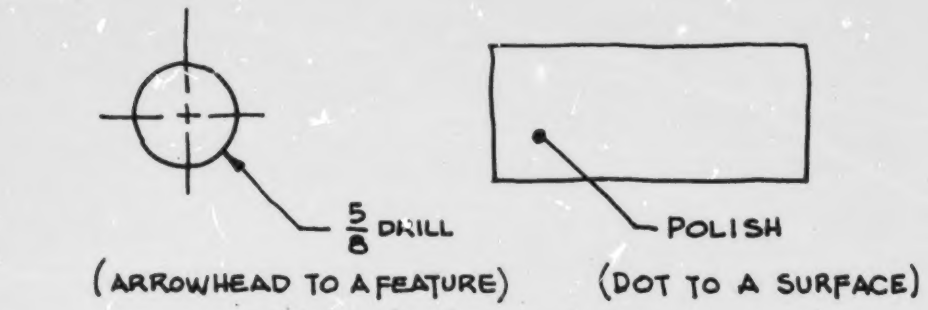


FIG 2-5

Dimension lines are the same thickness as extension lines and have arrowheads at each end to show where your measurement started and where it ends. Dimension lines should have a break in them so the measurement (dimension) you make can be shown. Extension and dimension lines are like Siamese twins; they are attached, and one can't be there without the other.

FIG. 2-6

Leader Lines are used to move the craftsman's eye from a surface, edge, or feature to a note. The leader is drawn at a 60°, 45°, or 30° angle; it is the same thickness as a dimension or extension line. It will have an arrowhead at the end to point out a feature or edge, and a dot to point out a surface. The reason you use leaders and notes on drawings is to prevent confusion by having too many dimensions in one area, having dimensions on the object itself, or having to break up information.



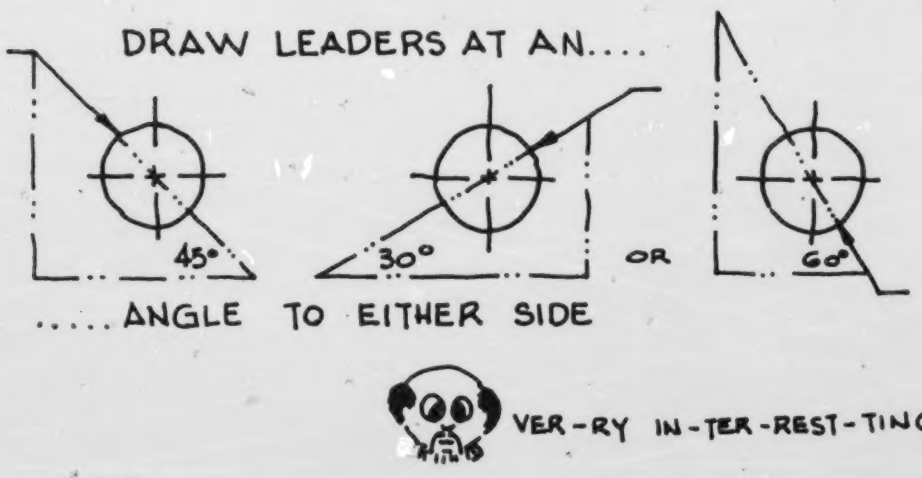


FIG. 2-7

HOW DO YOU USE THEM?

The usage of extension, dimension, and leaders varies, but the golden rule of drafting is "make it clear, but to the point". You don't want to give too little or too much information.

Here are some examples of how to use extension, dimension and leader lines.

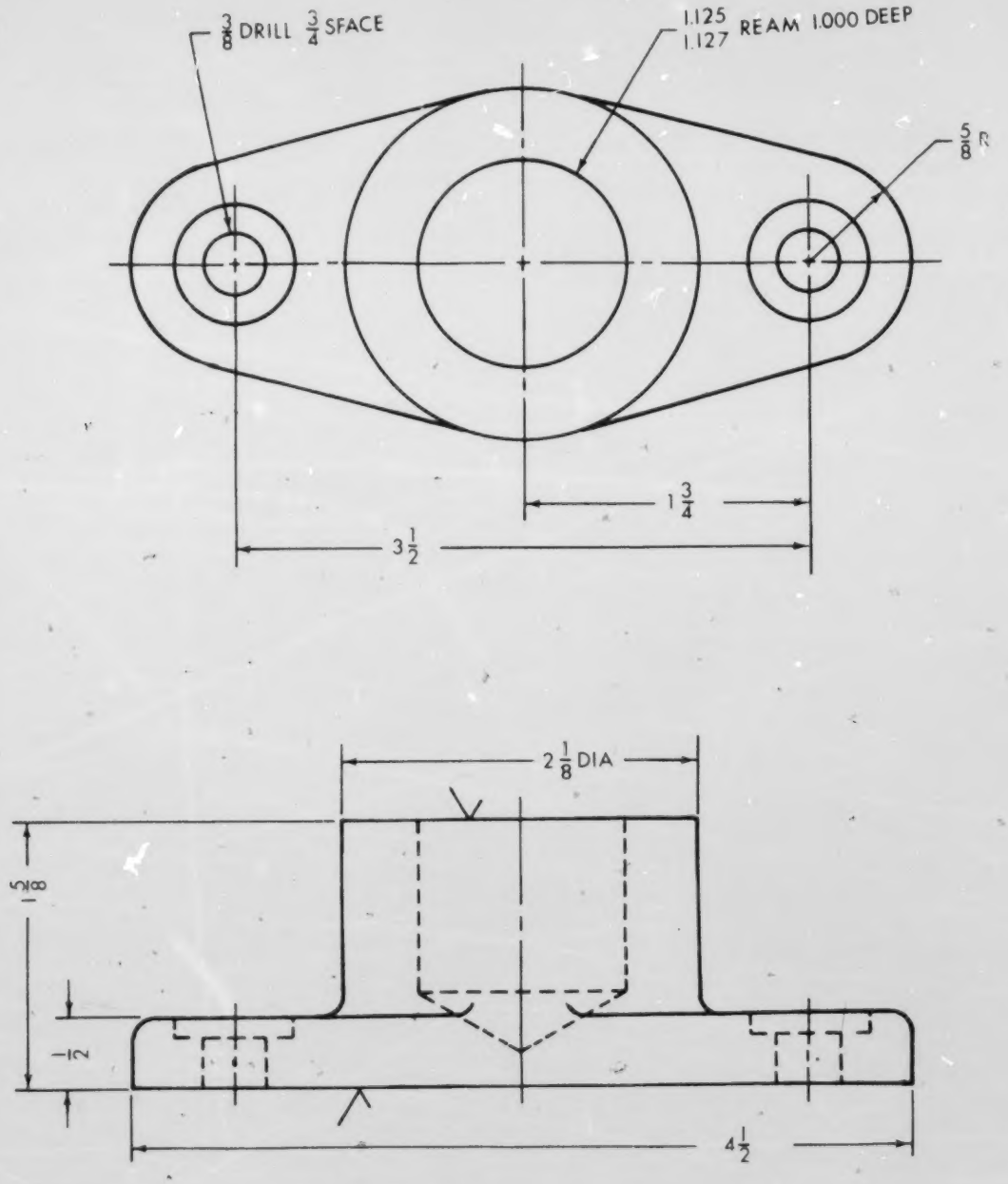


FIG. 2-8A

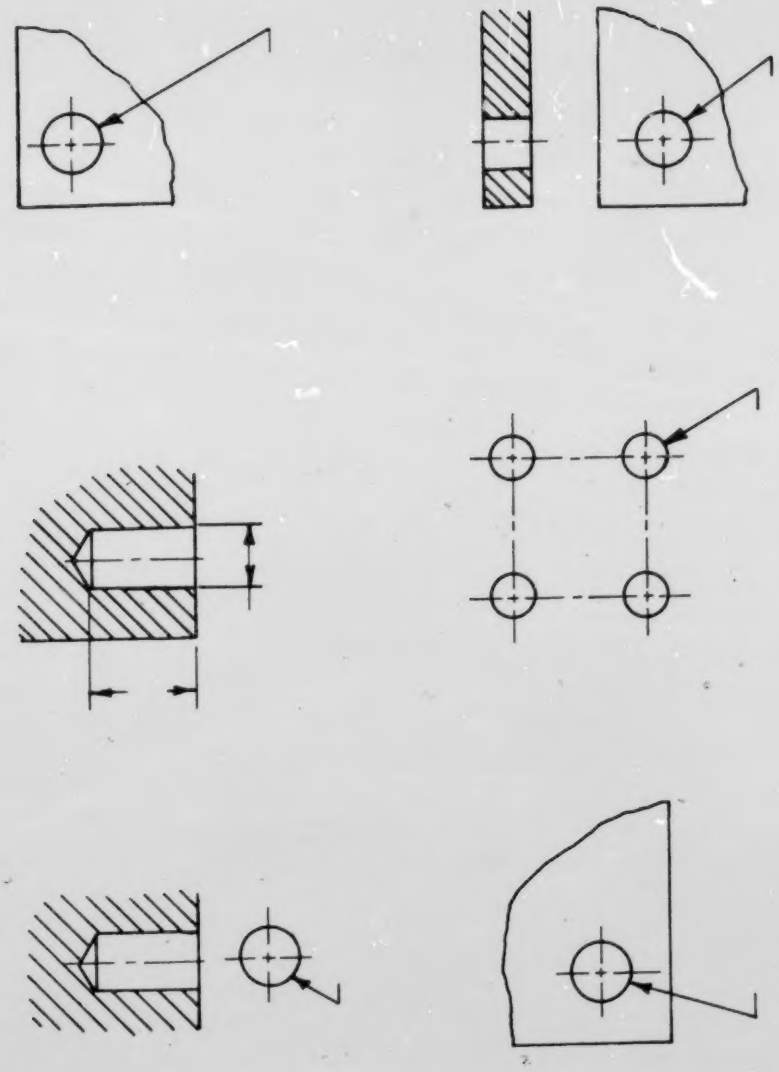


FIG. 2-8B

TAKE NOTE!

The notes you will be adding at the end of the leader line are called *local notes.* This is because they are bringing to your attention an operation to be performed at the *exact location* where the arrow or dot is pointing.

FIG. 2-9

If there are a few places where the exact same operation must be performed, you may use one local note (to describe the operation) followed by the number of places the operation is to be performed. Remember the golden rule of drafting! Don't use this method if the machinist will have difficulty in finding the location of a machine operation.

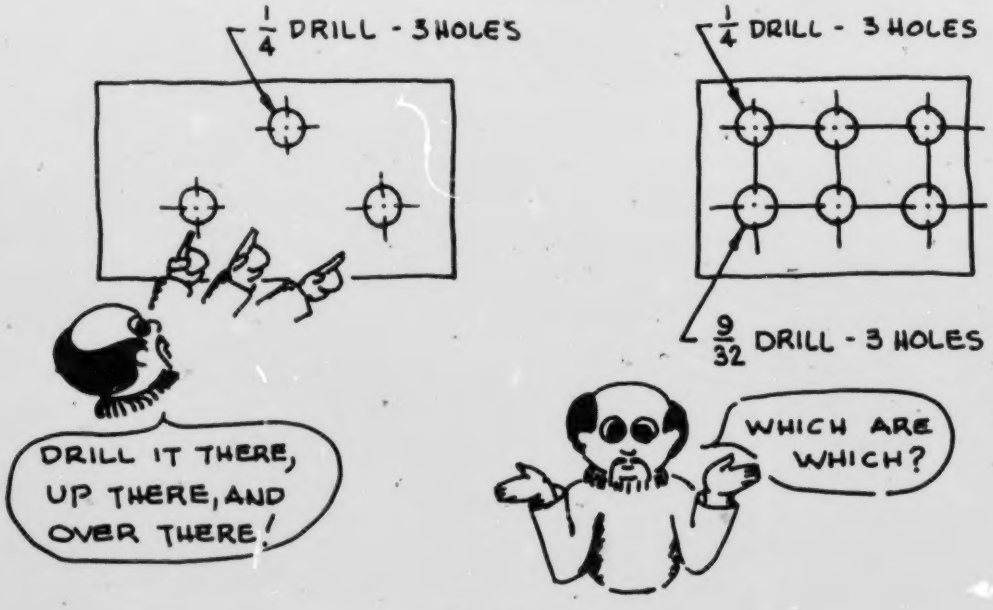


FIG. 2-10

If there is more than one operation to be performed at the same location, they can all be described in one local note. When describing the operations, make sure you list them in the order they will be performed.

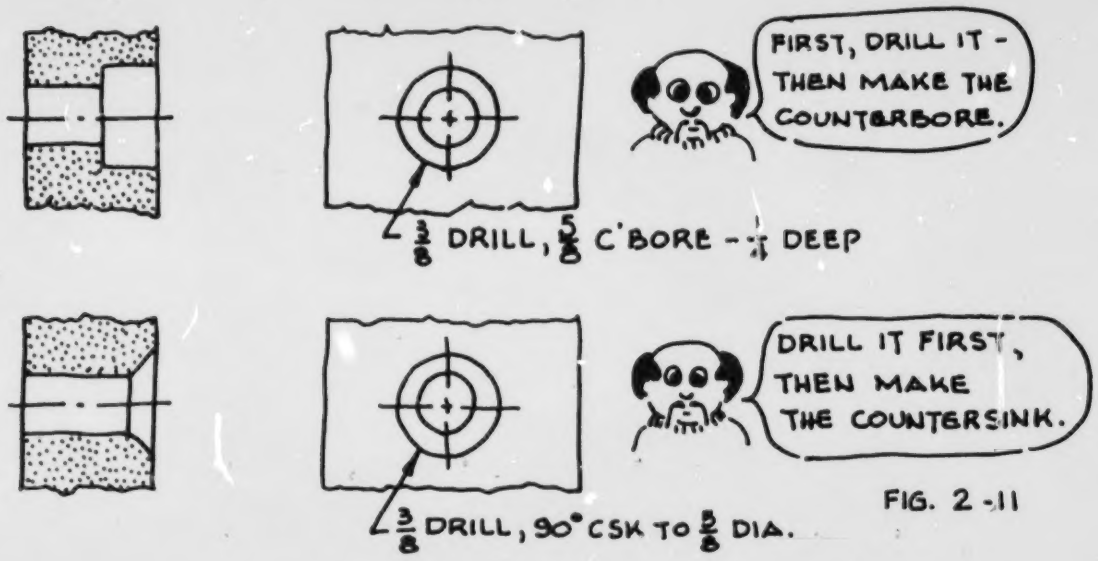

FIRST, DRILL IT - THEN MAKE THE COUNTERBORE.
3/8 DRILL, 5/8 C'BORE - 1/4 DEEP
DRILL IT FIRST, THEN MAKE THE COUNTERSINK.
3/8 DRILL, 90° CSK TO 5/8 DIA.

FIG. 2-11

General Notes are used to explain an operation or procedure which is used frequently or continually during the manufacturing of a machine part. They are also used to explain what will occur before or after the machinist has worked on the part. Basically, anything which applies to the manufacturing of a part (as a whole), its assembly with other parts, the testing of a part and other critical information on the part would go into a general note.

GENERAL NOTES:

1. ALL FILLETS AND ROUNDS 1/8 R.
2. FINISH ALL OVER
3. HEAT TREAT

FIG. 2-12

When listing general notes on a drawing, they should be placed above or to the left of the title block.

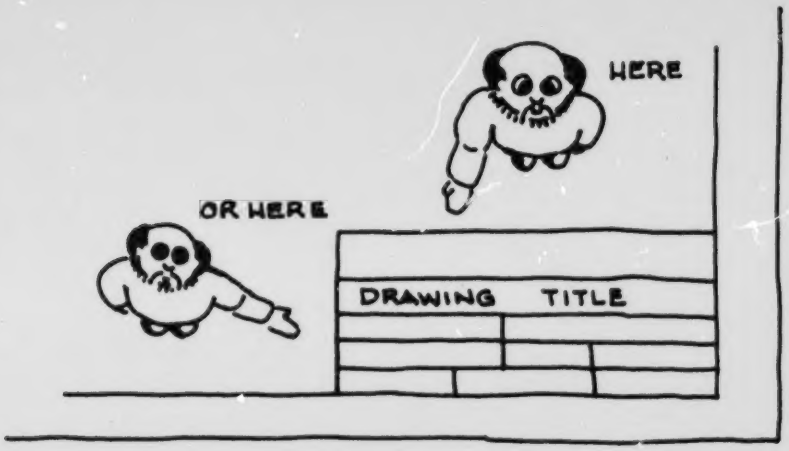


FIG. 2-13

METRIC OR ENGLISH?

Should you use the Metric or the English system to dimension your drawing? If you are used to answering questions the way you think the teacher wants them answered, you'd say "The Metric system!" If you're not that good at out-guessing the teacher, you'd say "The English system, 'cause this is America!" The problem is they are both wrong.

Stop scratching your head, it's a trick question. Since the United states is in the process of switching to the Metric system, ISO and ANSI (remember what they stand for?) developed a "dual system" -- using millimeters *and* inches.

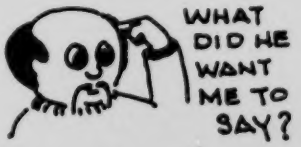


FIG 2-14

If you do go to work for a company using the English System exclusively, you should know something about how it is used. Most of the following standards apply to the English measurement system, whether it is used alone or with the Metric system.

If you are working with the English system only, you may use fractional dimensions. Whenever you use a fractional dimension, make sure you have reduced it.

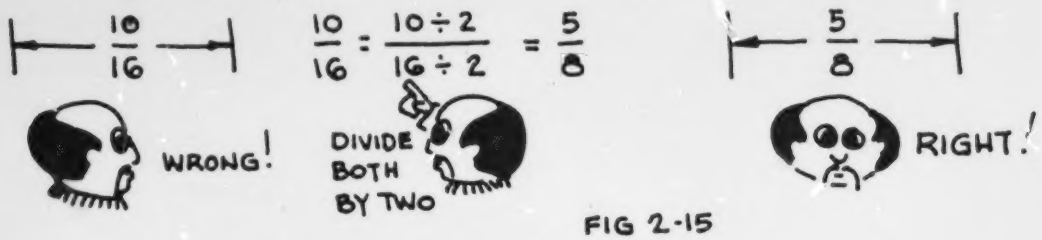


FIG 2-15

Whenever possible change all fractions to decimals. It is easier to convert a decimal to millimeters than a fraction.

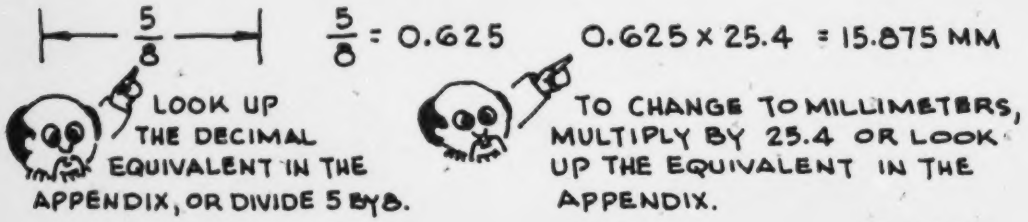

5/8
5/8 = 0.625
0.625 x 25.4 = 15.875 MM
LOOK UP THE DECIMAL EQUIVALENT IN THE APPENDIX, OR DIVIDE 5 BY 8.
TO CHANGE TO MILLIMETERS, MULTIPLY BY 25.4 OR LOOK UP THE EQUIVALENT IN THE APPENDIX.

FIG. 2-16

Whenever you are dimensioning large objects, don't mix feet and inches on machine drawings. This standard varies with each industry and each area of drafting. For now, dimension objects *72" and below in inches only* on machine drawings.

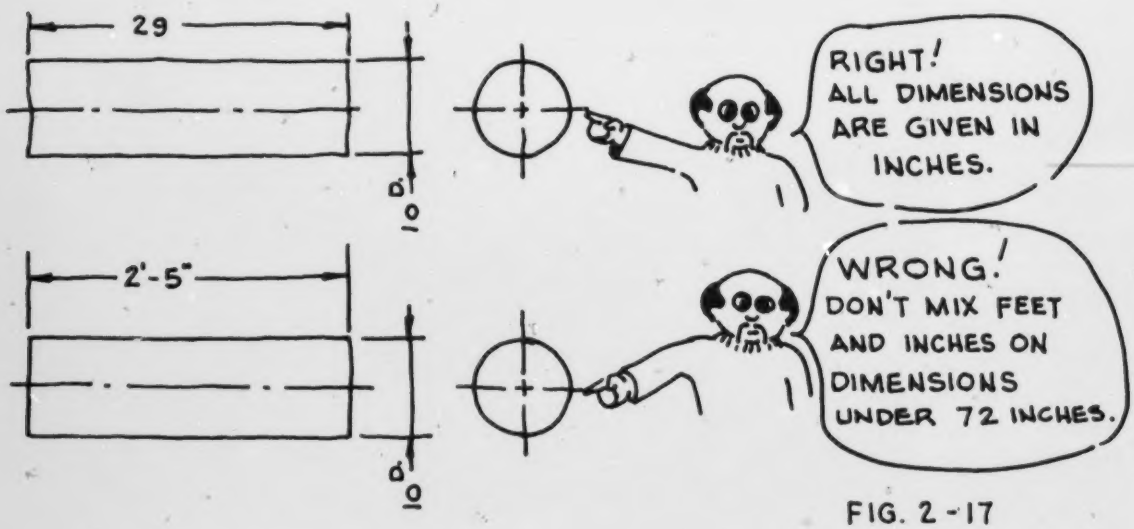


FIG. 2-17

Architectural drawings will use inches only below 12" They would use either feet and inches, or just inches for 24" and below. Above 24" would be expressed in feet and inches.

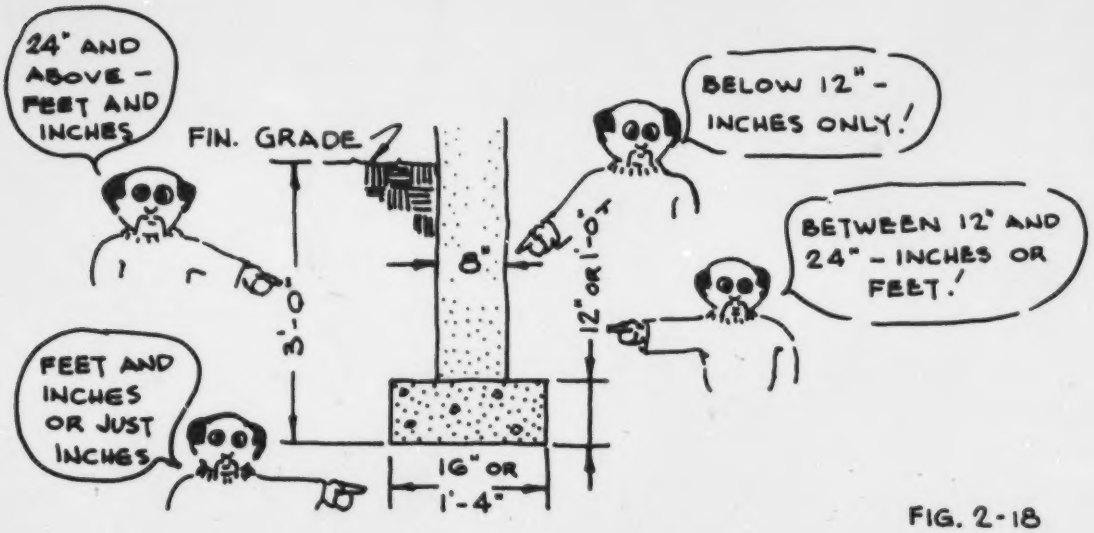

24" AND ABOVE - FEET AND INCHES
FIN. GRADE
BELOW 12" - INCHES ONLY!
BETWEEN 12" AND 24" - INCHES OR FEET.
FEET AND INCHES OR JUST INCHES
3'-0"
8"
12" OR 1'-0"
16" OR 1'-4"

FIG. 2-18

Civil Drafting will use either feet and inches for dimensions 12" and above, or feet and decimal equivalents of a foot (parts of foot expressed in decimal form).

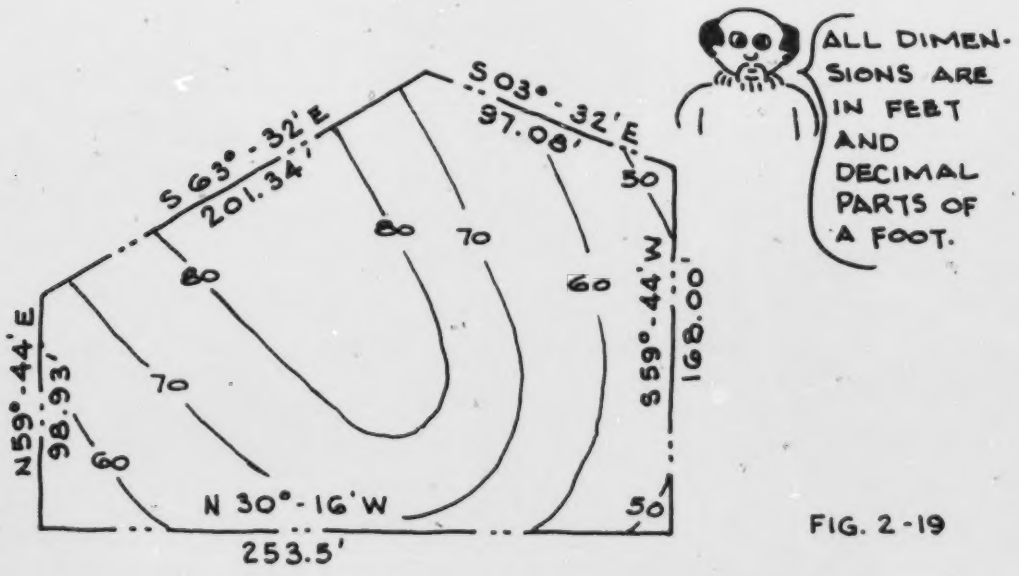


FIG. 2-19

If your drawing has both English and Metric dimensions, you must use (mm) millimeter symbols if most of the dimensions are in inches, or the inch symbol (IN) if most of the dimensions are in millimeters. A note must be added to explain that, unless otherwise specified, all dimensions are in inches or millimeters, as the case may be. (What type of note is this?)

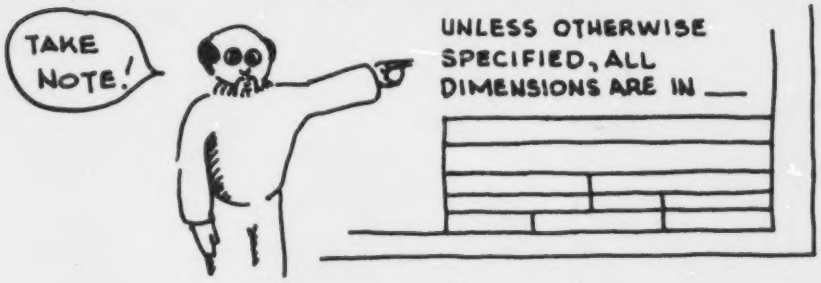


FIG. 2-20

If you use only millimeters *or* inches, you should not use the millimeter (mm) symbol or the inch symbol (IN or "). You should, however, note which dimensioning system you are using. The only exception to this rule is when there is a chance of confusion (Golden Rule).

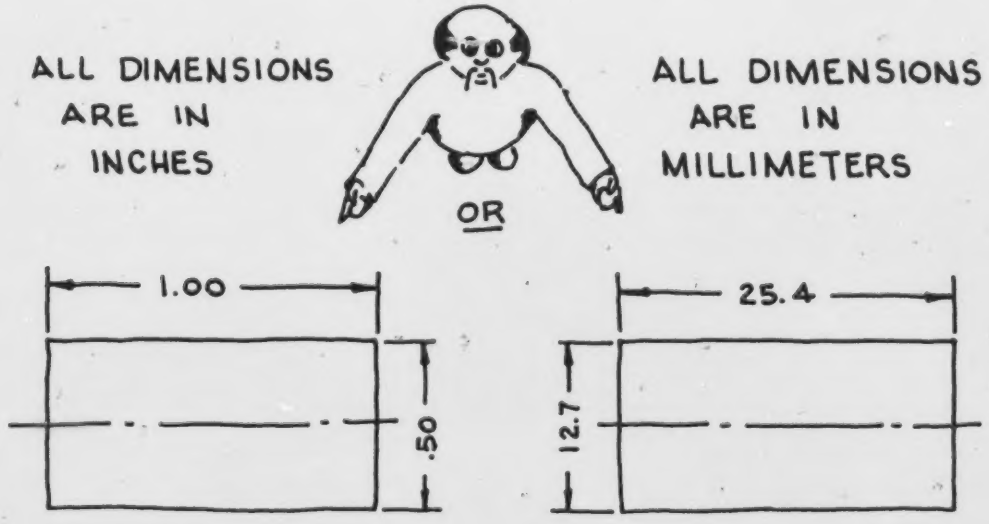


FIG. 2-21

When you combine systems completely (dual dimensioning), *all* dimensions will be given in *inches and millimeters.* Inch dimensions will be given as decimals -- millimeter dimensions are always given as decimals anyway. The reason for using both systems is to prevent a possible error in conversion by the machinist. Also, until conversion is complete, there will be some manufacturers and/or customers who are still using the English system.

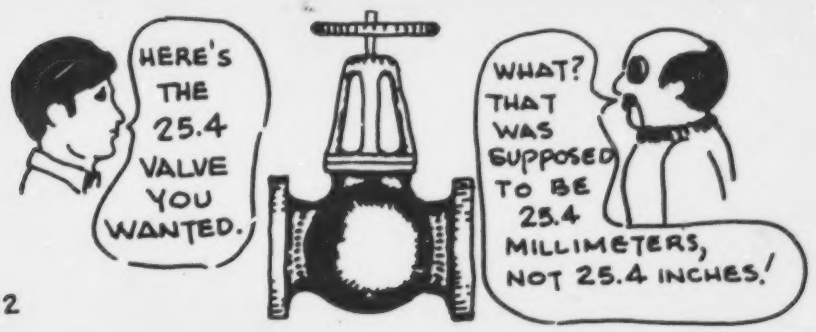


FIG 2-22

There are two accepted methods of expressing dual dimensions -- position and bracket methods. In the position method, the Metric dimension is placed above the dimension line and the inch dimension is placed below it. The two dimensions can be reversed, as long as it is explained in a note which dimension is which, and you are consistent throughout the drawing.

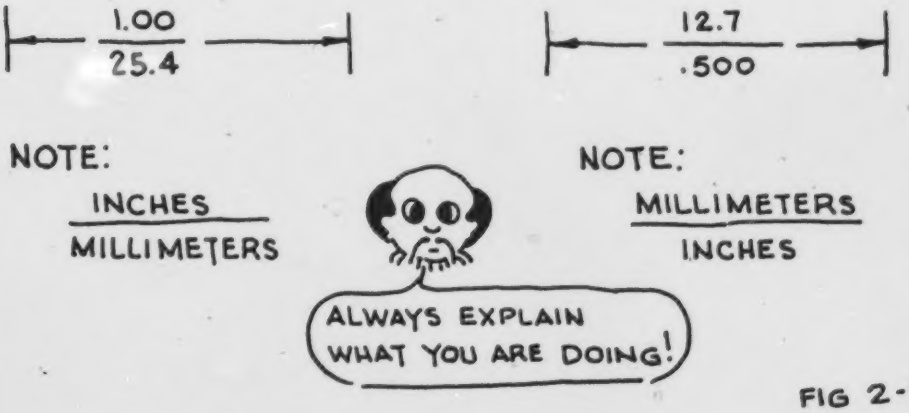
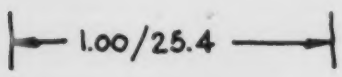
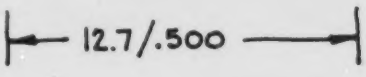


FIG 2-23

An alternate of the position method is to give the Metric dimension followed by a slash and the inch dimension. Again, the dimensions can be reversed.



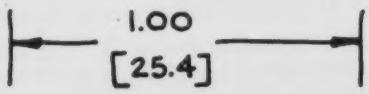
NOTE:
INCHES/MILLIMETERS



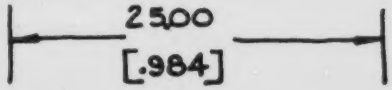
NOTE:
MILLIMETERS/INCHES

FIG. 2-24

In the bracket method, the inch dimension (decimal form) is given first. The Metric dimension is placed under the inch dimension and then enclosed by brackets. This process may be reversed, but in either case it must be noted which dimension is in brackets.



NOTE:
ALL DIMENSIONS
IN [ ] ARE
MILLIMETERS



NOTE:
ALL DIMENSIONS
IN [ ] ARE
INCHES.

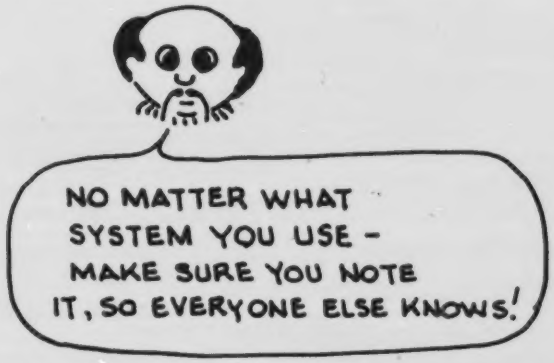


FIG 2-25

WHAT'S YOUR ANGLE?

Angles are measured in degrees (°) minutes (′), and seconds (″); or in decimal parts of a degree. This holds true whether you are working with the English or the Metric system.

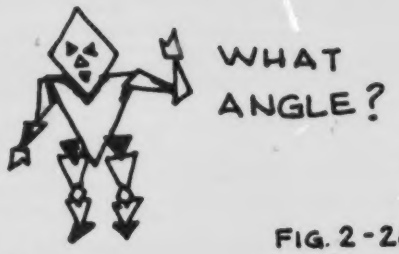
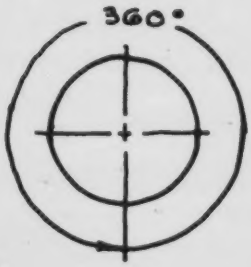


FIG. 2-26

There are 360° in a circle; each degree is made up of 60′; each minute is made up of 60″ (seconds). Keep this in mind while you are working with angles!



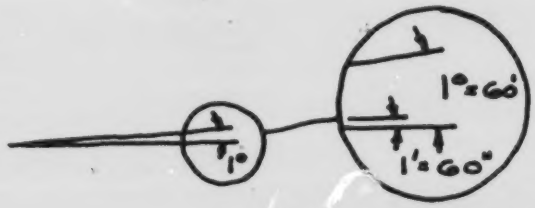


FIG. 2-27

If you are going to add two or more angles together and the total number of seconds equals 60 or more, subtract 60 or a multiple of 60 until the remainder (what's left) is under 60. Then add the number of 60's you have subtracted to the total number of minutes. Follow the same procedure with the minutes and add the number of 60's to the total number of degrees.

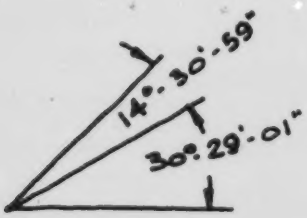


FIG. 2-28

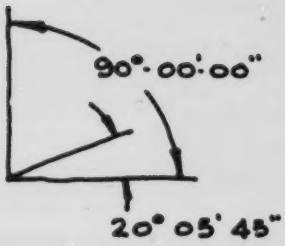
	30°	29′	01″
+	14°	30′	59″
	44°	59′	60″

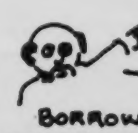
44°+1°=45° 1′+59′=60′ OR 1° EQUALS 1′

ANSWER = 45° 00′ 00″

If you are going to subtract one angle from another, and the number of seconds you are subtracting is greater than the number of seconds you are subtracting from, you will need to borrow a minute. *Remember,* one minute equals 60 seconds. Add 60 to the number of seconds and finish your subtraction. The same holds true for the minute

column — you will borrow from the number of degrees, however. *Remember*, one degree equals 60 minutes.





$$\begin{array}{rrrr} & 90^\circ & 00' & 00'' \\ - & 20^\circ & 05' & 45'' \\ \hline \end{array}$$

BORROW 1° WHICH EQUALS 60' — THEN BORROW 1' WHICH EQUALS 60"

$$\begin{array}{rrrr} & 89^\circ & 59' & 60'' \\ - & 20^\circ & 05' & 45'' \\ \hline \text{ANSWER} = & 69^\circ & 54' & 15'' \end{array}$$

FIG. 2-29

When you are dimensioning an angle, you can use extension lines or the object itself to show the angle you are dimensioning. It is better to keep *all* dimensions off the object, including angular dimensions, whenever possible. The dimension lines used for angles will be arcs (parts of a circle), drawn with the apex (pointed end) of the angle as the center.

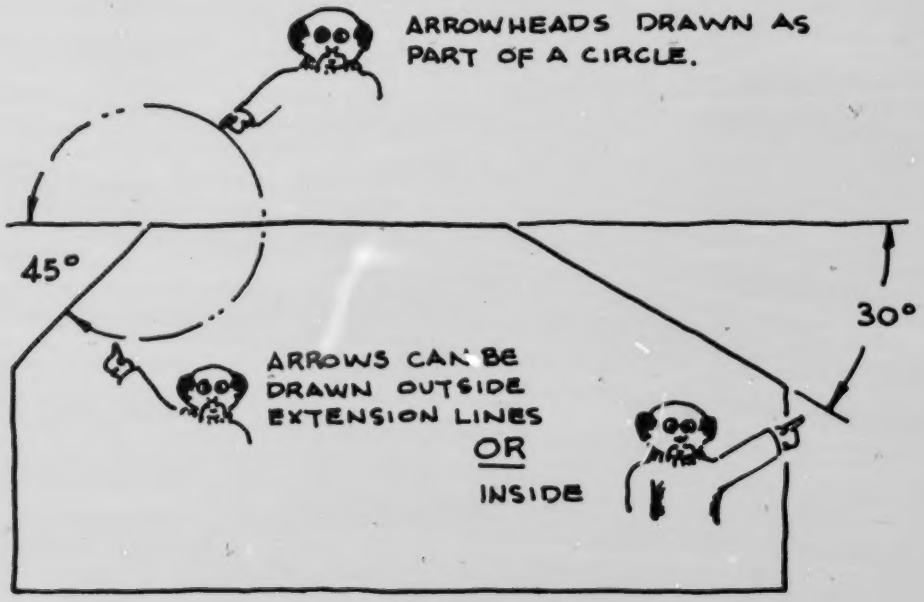


FIG. 2-30

You should now know the *basic* dimensioning symbols and methods of expressing dimensions. Let's take time out to review this information before going further.

THINK ABOUT IT

Are the draftsmen and machinists the only people using "dual measurements"? (Hint: Have you eaten any cereal lately?)

There are a number of professionals who have their own "language", such as doctors. Why was it necessary to develop "professional jargon"? Why is it necessary to learn the language of a profession?

Since the symbols of drafting are used to describe an object, and the drawing is done to scale, why do you need dimensions and notes?

Where did drafting symbols and standards come from? Can you see logic behind the forms that were chosen?

EXERCISE YOUR BRAIN

What are dimension lines and what do they look like?

What are extension lines and what do they look like?

Explain what relationship (connection) extension and dimension lines have with each other.

What *must* you check before using fractional dimensioning on a drawing?

Whenever possible, what *should* you do to fractional dimensions before using them on a drawing?

What do mm and IN mean ?

When you are using both English and Metric dimensions (mixed) on a drawing, what should you do to prevent confusion? (Hint: This is not dual dimensioning.)

What is dual dimensioning?

What methods are used to specify dimensions and prevent confusion when using dual dimensioning?

What units of measurement are used to dimension angles?

How do angular dimensioning practices differ from linear dimensioning practices?

UNIT III: DIMENSIONING –– SIZE, SHAPE, LOCATION

WHERE? WHAT?

Remember, dimensions help to tell the story of a machine part. In telling the story, you must answer the same questions a good reporter answers in his articles –– who, what, where, when, how, and why.

The question "What is to be done?" is answered in the notes and symbols used on a drawing. The question "How is it to be done?" is answered in what is called a "parts production program or plan". We will get into this in a later chapter. The question "Why is it done?" or "Why is it done that way?" is usually answered by experience and knowledge of machine processes. Again, this will be covered in a later chapter. The questions "Who will do it?" and "When will it be done?" are usually answered by both notes and in the production program. Also, you will need to have a good working knowledge of machinery and what it can do. The questions "Where is an operation to be performed?", "What size is the object?", and "What shape is the object?" are answered by size, shape, and location dimensioning.

HOW BIG IS IT?

The size of an object is important to you, the machinist, and the customer. It's important to you, so you can design and draw the object; it's important to the machinist so that he can make the part; and it's important to the customer so he can figure the cost and his needs.

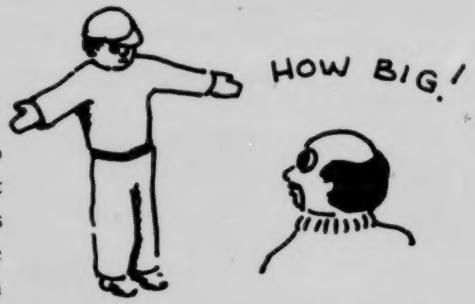
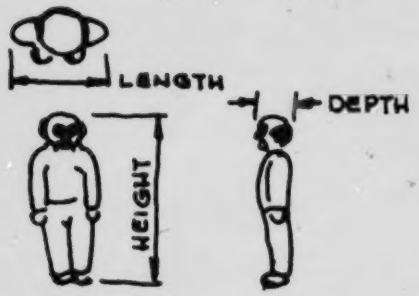


FIG. 3-1

When you are trying to describe the size of an object, the basic dimensions you will need are the height, width (or length), depth (or width), and the scale of the drawing. These *overall dimensions* should be placed between the views or next to the view that shows the dimension in *true length*.



DIMENSIONS CAN BE SEEN IN TWO VIEWS AND THEY SHOULD BE NOTED BETWEEN THOSE TWO VIEWS.

FIG. 3-2

As you can see, in 3rd angle projection, the height is given between the front and side views; the width is given between the top and front views; and the depth is given between the top and right side view.

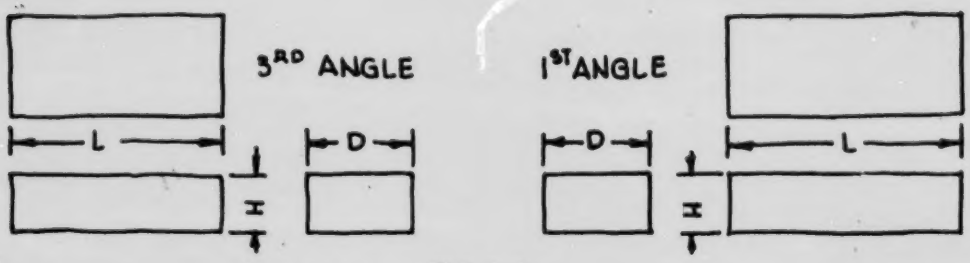


FIG. 3-3

The overall dimensions should be outside all the other dimensions. This is for two reasons: (1) To prevent confusion caused by extension and dimension lines crossing, and (2) to aid the machinist in choosing the stock size by having the dimensions readily visible.

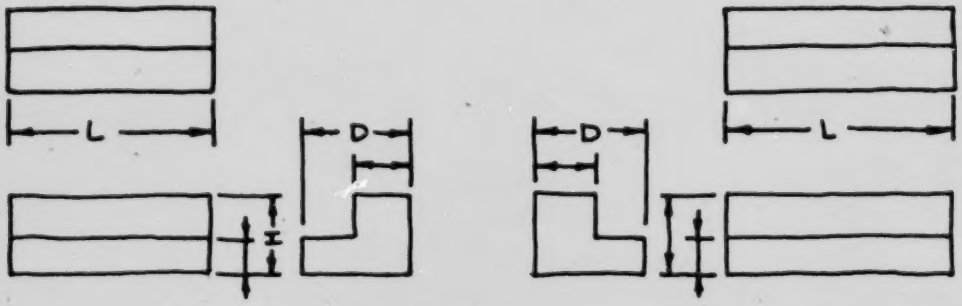
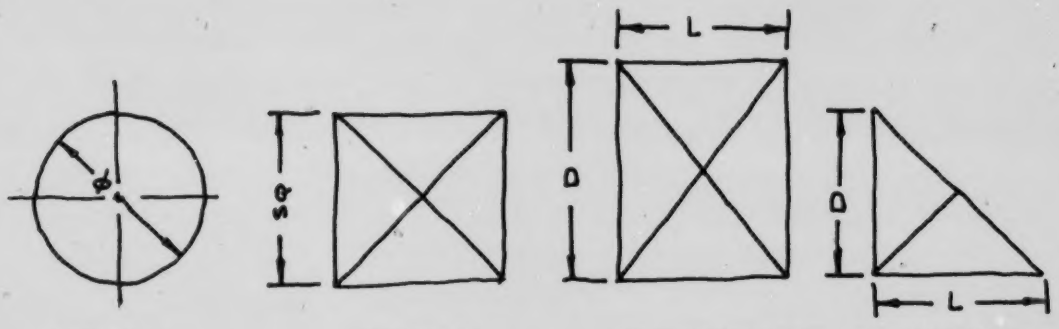


FIG. 3.4

WHAT A SHAPE!

Under normal circumstances, you will be able to *see* the shape of the object in three-view orthographic projection -- *but*, you won't always be able to. Also, when you don't use all three views, in order to save time and space, you won't always be able to show the shape of the object. When these conditions occur, you're going to have to rely on dimensions and notes to completely describe the shape of the object. Consider the following examples:

FIG. 3·5



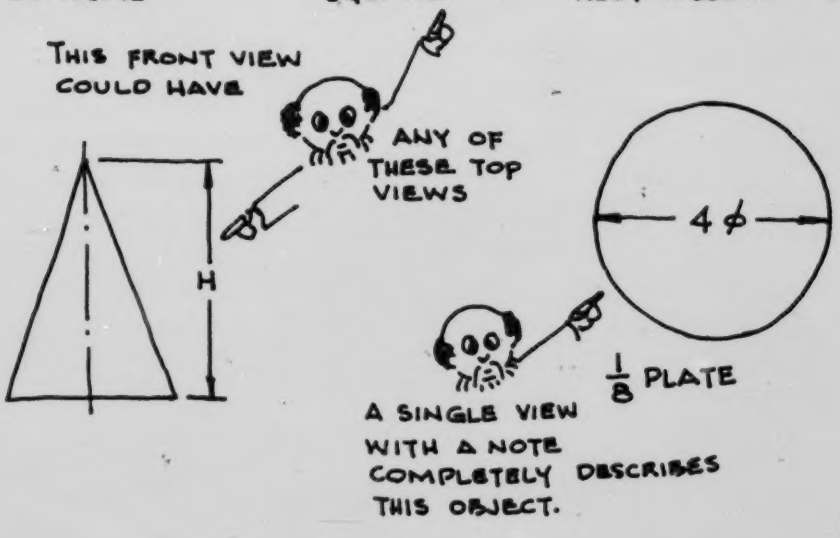
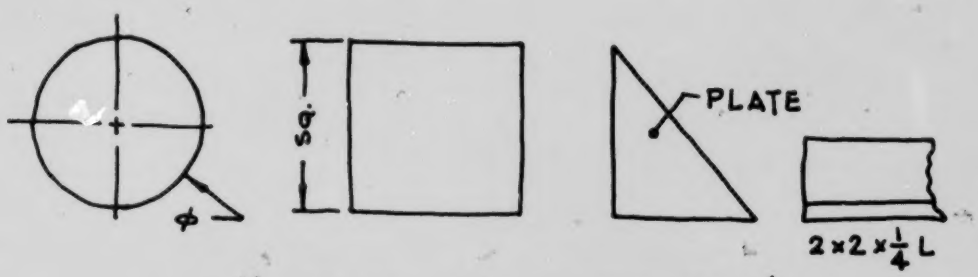


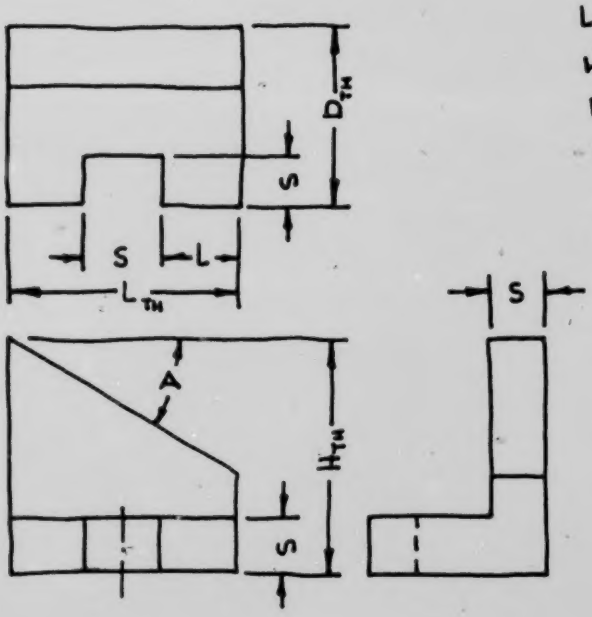
FIG 3·6

Whenever you run into a situation like this, it is easier if you give the shape of the object in the dimension.



NOTES CAN DESCRIBE THE OBJECT'S SHAPE AND SAVE DRAWING TIME. FIG. 3·7

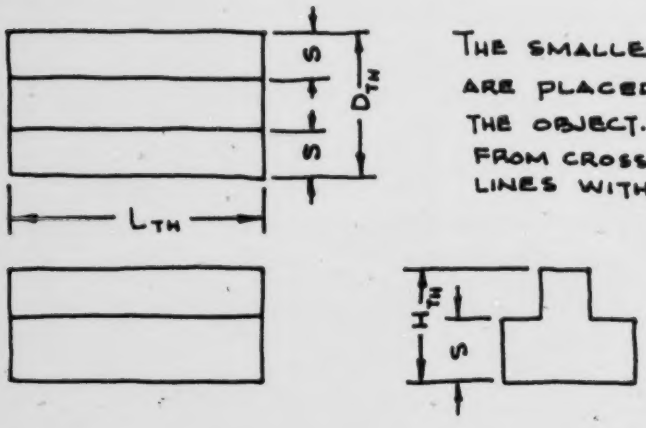
Besides these special cases, there are other practices which must be followed to describe the object's shape. The dimensions should be placed next to the view that best shows the shape to be dimensioned.



LTH = LENGTH
HTH = HEIGHT
DTH = DEPTH
L = LOCATION DIMENSION
S = SIZE DIMENSION
A = ANGULAR DIMENSION

DIMENSIONS SHOULD BE PLACED IN THE VIEW WHICH BEST SHOWS THE OBJECTS SHAPE. FIG. 3·8

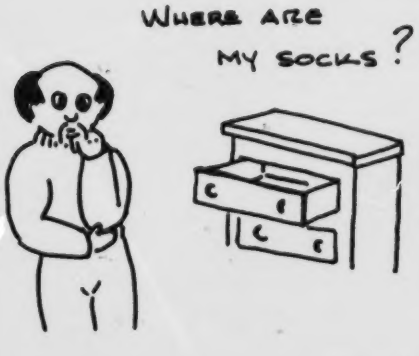
The smallest dimension should be placed nearest the view being dimensioned.



THE SMALLEST DIMENSIONS ARE PLACED CLOSEST TO THE OBJECT. THIS IS TO KEEP FROM CROSSING OVER DIMENSION LINES WITH EXTENSION LINES.

FIG. 3·9

WHERE IS IT?



Location dimensions do exactly what the name implies ––locate something. These dimensions will tell you *where* –– where to drill, where to make a cut, etc. There are three methods used to locate an operation on a drawing: (1) rectangular coordinates, (2) polar coordinates and (3) chordal dimensioning.

FIG. 3·10

Rectangular coordinates, or the rectangular dimensioning system, is a "method of indicating distance, location, and size by means of linear dimensions measured parallel or perpendicular to reference axes or planes that are perpendicular to each other". [1] Don't run, yet! All they are trying to say is: (1) you will have some dimensions running horizontal and some vertical; (2) there will be a surface or center line used as a reference,

[1] *Engineering Drawing and Design,* C. H. Jensen, McGraw–Hill Company of Canada, Ltd., 1968, page 65.

a starting point for all dimensions, running horizontal or vertical; and (3) there will be *two* reference planes (surfaces) or axes (center lines) used in the drawing. There can be more than two reference planes or axes on more complex objects.

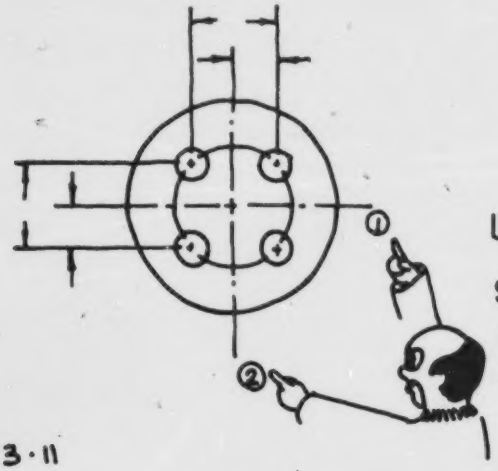

RECTANGULAR
COORDINATE
DIMENSIONING
USE TWO REFERENCE LINES - 90° APART - TO DIMENSION FROM. THE CENTERLINES - ONE VERTICAL AND ONE HORIZONTAL - ARE USED AS REFERENCE LINES.
①
②

FIG. 3·11

Polar or angular dimensioning is "a method of indicating the position of a point, line, or surface by means of a linear dimension and an angle, other than 90°, that is implied by vertical and horizontal center lines". [2] Basically, polar dimensioning locates something by giving how many degrees, minutes, and seconds (angular measurement) it is from two intersecting (crossing) center lines (one horizontal, one vertical) and how far it is from that intersection (linear measurement.) Again, you can also use a surface as a reference rather than the center lines. 90° angles don't have to be dimensioned because the point you are dimensioning would fall on the horizontal or vertical center line, and by definition they are 90° apart.

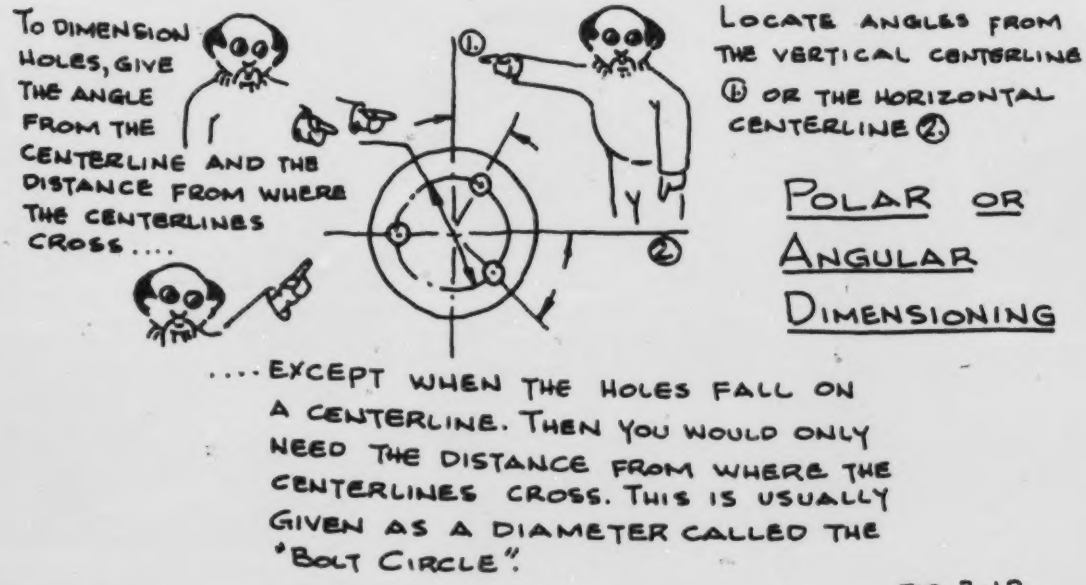
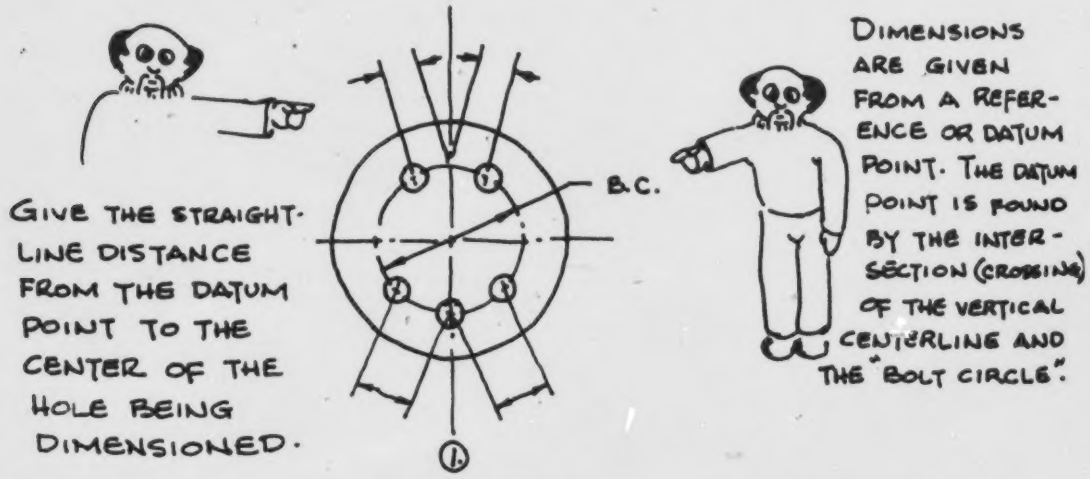

TO DIMENSION HOLES, GIVE THE ANGLE FROM THE CENTERLINE AND THE DISTANCE FROM WHERE THE CENTERLINES CROSS....
LOCATE ANGLES FROM THE VERTICAL CENTERLINE ① OR THE HORIZONTAL CENTERLINE ②
POLAR OR ANGULAR DIMENSIONING
....EXCEPT WHEN THE HOLES FALL ON A CENTERLINE. THEN YOU WOULD ONLY NEED THE DISTANCE FROM WHERE THE CENTERLINES CROSS. THIS IS USUALLY GIVEN AS A DIAMETER CALLED THE "BOLT CIRCLE".
①
②

FIG. 3·12

[2] IBID, page 65

Chordal dimensioning is "used for the spacing of points on the circumference of a circle relative to a datum, where manufacturing methods indicate that this will be more convenient". [3] The first thing you should take notice of is that *manufacturing methods* can force you to use one dimensioning system rather than another —— a point that will be explained further in another unit.


GIVE THE STRAIGHT-LINE DISTANCE FROM THE DATUM POINT TO THE CENTER OF THE HOLE BEING DIMENSIONED.
B.C.
DIMENSIONS ARE GIVEN FROM A REFERENCE OR DATUM POINT. THE DATUM POINT IS FOUND BY THE INTERSECTION (CROSSING) OF THE VERTICAL CENTERLINE AND THE "BOLT CIRCLE".

CHORDAL DIMENSIONING

FIG 3-13

An arc is part of a circle, and the straight line joining the beginning of the arc and the end of the arc is called a chord. In chordal dimensioning, you are giving the measured linear (straight line) distance from the center line (datum) to the center of the operation to be performed. This is done *only* when you have a bolt circle or other circular feature to dimension.

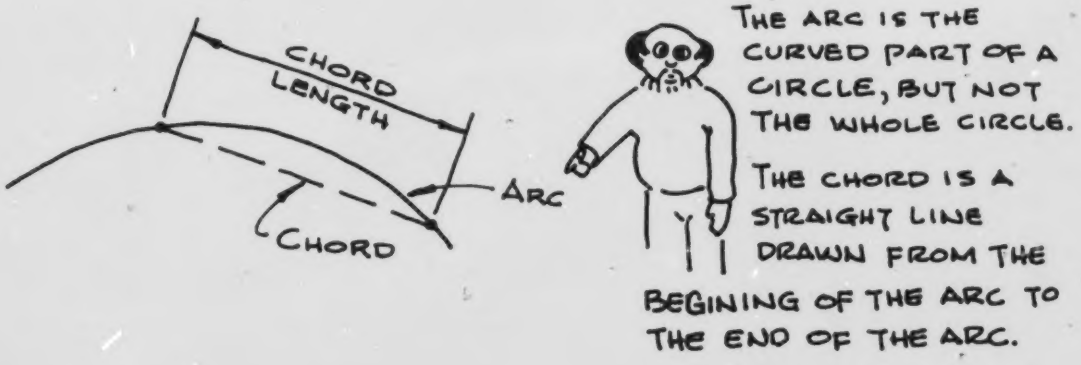
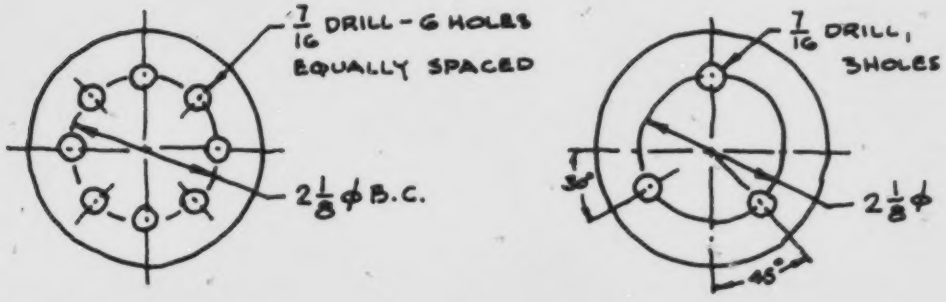

CHORD LENGTH
ARC
CHORD
THE ARC IS THE CURVED PART OF A CIRCLE, BUT NOT THE WHOLE CIRCLE.
THE CHORD IS A STRAIGHT LINE DRAWN FROM THE BEGINING OF THE ARC TO THE END OF THE ARC.

FIG 3-14

[3] IBID, page 66.

Now that we have talked about size, shape, and location dimensions, take a look at the following examples and see if you understand why they were dimensioned the way they were.



6 HOLES; 360° IN A CIRCLE

$6\overline{)360}^{60}$ THE HOLES ARE 60° APART

FIG. 3·13

THINK ABOUT IT!

Why do you think size is important to a manufacturing company, whether buying or selling equipment?

European countries use 1st angle projection, and the United States uses 3rd angle projection. What are the differences, and why do you think these differences occurred?

Why is it so necessary to make things clear to the machinist? *Remember:* The machinist isn't stupid; he is a trained craftsman with an equal degree of intelligence as you, but with different skills and interests.

Why would you need so many different methods to locate points, lines, and/or surfaces? Can you see a logical progression in the methods used to dimension a location?

Why do you think manufacturing methods would control a method of dimensioning?

EXERCISE YOUR BRAIN!

What questions do size, shape, and location dimensions answer?

Where should the overall dimensions be placed on a drawing?

Under what conditions should you note the size of an object?

What three methods of location dimensions have been explained to you?

How do the location dimension systems differ?

Give a brief explanation of each.

What is an arc?

What is a chord?

WHAT NOW?

FIG. 4-1

You now have an idea of how to begin dimensioning an object, but you haven't scratched the surface yet. The basic dimensioning practices and systems described in Units II and III will lead you into this and following units.

The dimensioning systems we will be talking about in this unit will deal with *how* you will place dimensions of your drawing. How are you going to arrange the dimensions? How are they going to read? What are you going to base your decisions on?

READING REGULATIONS!

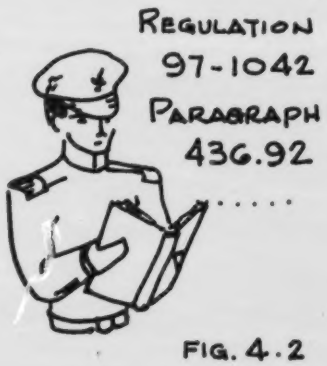


FIG. 4.2

Dimensions are placed in patterns, and a pattern is kept consistent throughout a drawing. These patterns were not haphazardly devised, but were developed after years of work in the profession; they are now standards. The first pattern we will discuss is "How should the dimensions be placed so they can be read easily?"

There are two acceptable answers to this question: (1) Dimensions should be placed so they can all be read from the bottom of the drawing; and (2) dimensions should be placed so they can be read from the bottom *and* the right-hand side of the drawing. The first system is called the *unidirectional system*, and the second is called the *aligned system*.

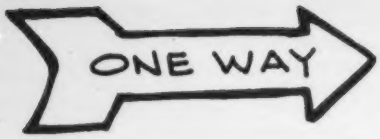
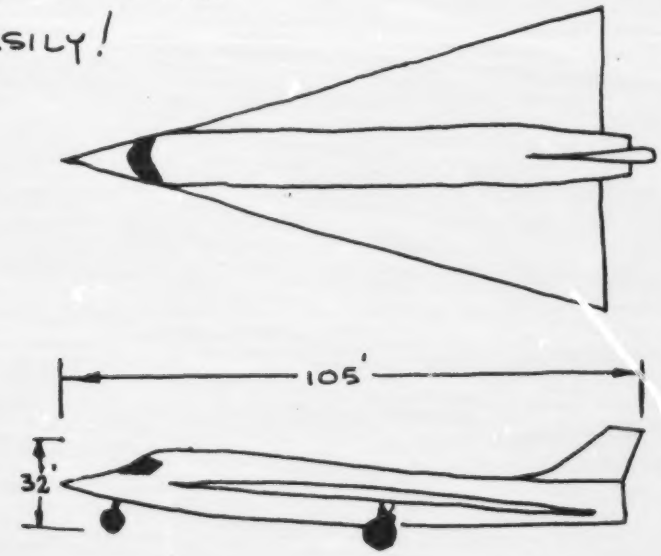


FIG. 4-3

The unidirectional system gets its name from "uni-," meaning one, and "direction," meaning way – *one way*. All dimensions must read from only one direction, and this is almost always from the bottom of the drawing. (The exceptions are limited to a few industries.) The unidirectional system is used in the aircraft, aerospace, automotive, and map-making industries. Any industry which uses large drawings that are hard to shift around would use the unidirectional system because it is easier to read.

INDUSTRIES USING LARGE DRAWINGS REQUIRE UNIDIRECTIONAL DIMENSIONS SO THEY CAN BE READ EASILY!



ALL DIMENSIONS SHOULD BE PLACED SO THEY CAN BE READ FROM THE BOTTOM OF THE PAPER!

FIG. 4-4

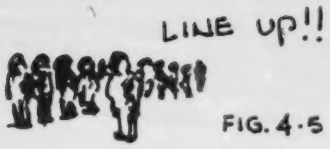


FIG. 4-5

The aligned system gets its name from the fact that the dimensions "line up" with the surface to be dimensioned. The dimensions are set up so they can be read from the bottom and the right-hand side.

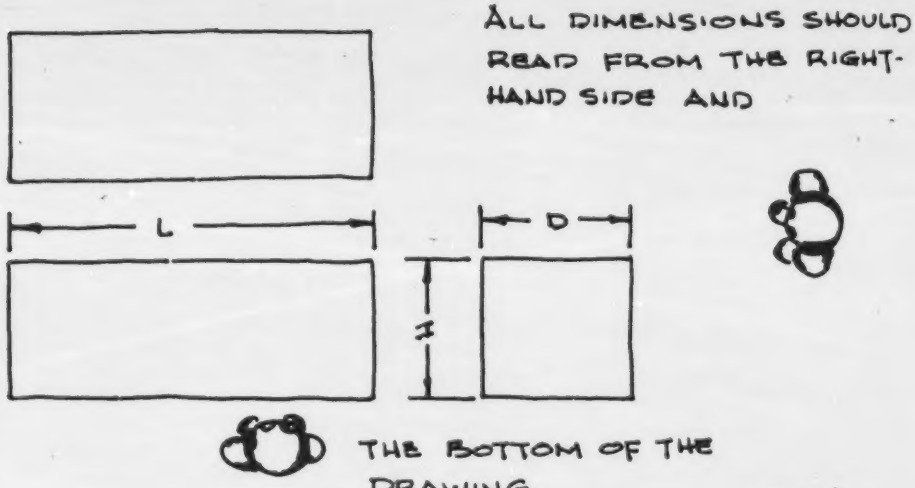


FIG. 4-6

This creates some problems, since some of the dimensions can't be read from the right if they are lined up with the surface. If you run into this problem or are in doubt about reading a dimension, move the dimension to the position you are sure can be read from the bottom or right side.

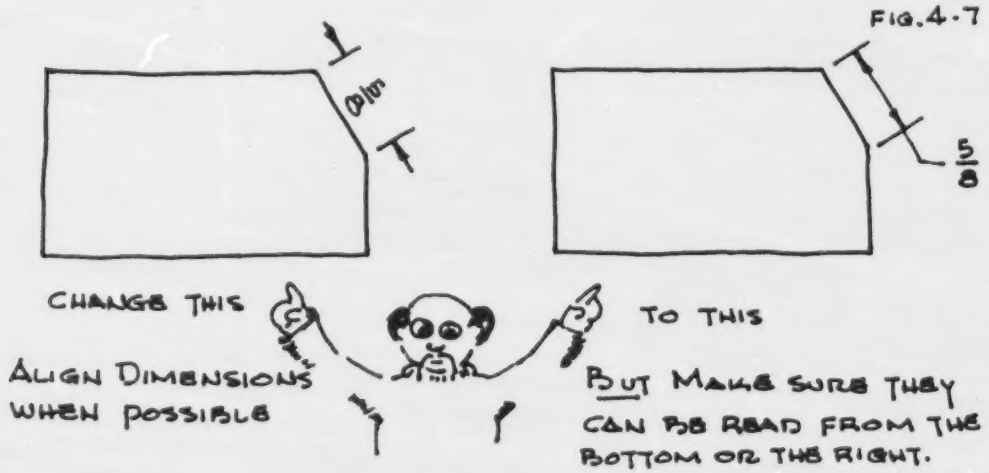


FIG. 4-7

When you are dimensioning arcs, angles, or circular objects, there are two areas which should be avoided. If a dimension falls in these areas, give complementary angle (subtract the angle you have from 90°) instead or change the position of the dimension.

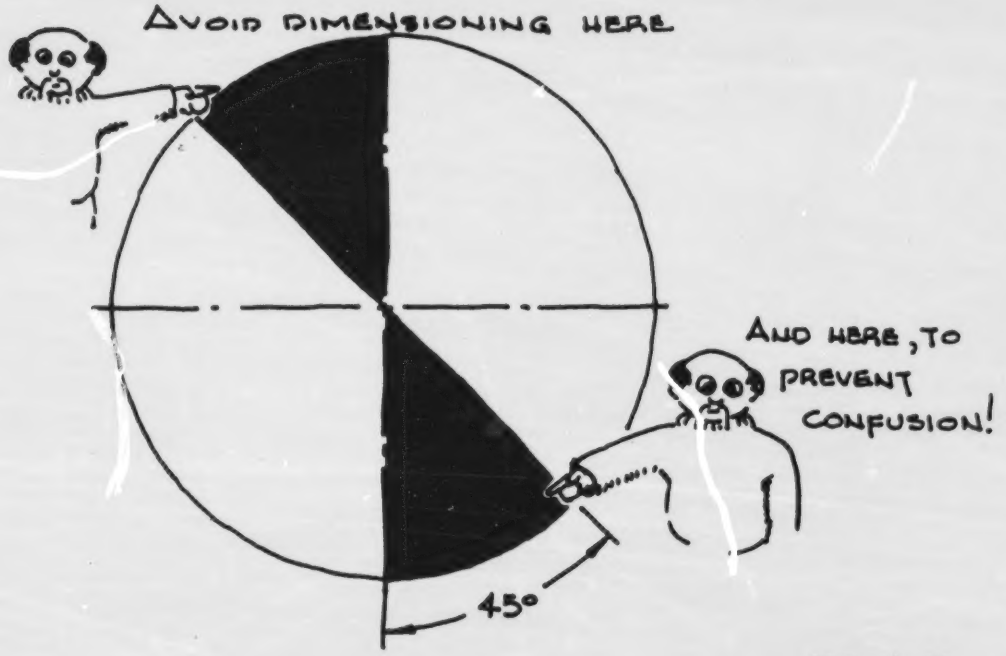


FIG. 4-8

BASED ON . . . . .

The arrangement and importance of dimensions (priority) are determined by *manufacturing methods* and *sequence of operations.* Sequence of operations means the order in which the machine operations take place. With a little knowledge of manufacturing methods and machine shop practices, recognizing the order in which the operations will be performed becomes fairly simple and logical. The two standard methods of presenting and choosing dimensions are chain dimensions and baseline dimensions.

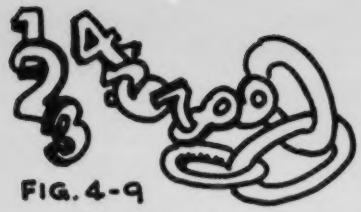


FIG. 4-9

CHAIN GANG

Chain dimensions are "linked", one after another, in a straight line. This method is simple and easy, since you don't worry about the sequence of operations. The machinist is left holding the bag, though, since he *has to worry* about the sequence of operations.

In chain dimensioning you would start at one edge of the object, working your way to the other edge and making sure you have located all center lines, machine operations, and important features or surfaces in between. *You would leave out one dimension in the chain* and give the overall dimension to prevent confusion.

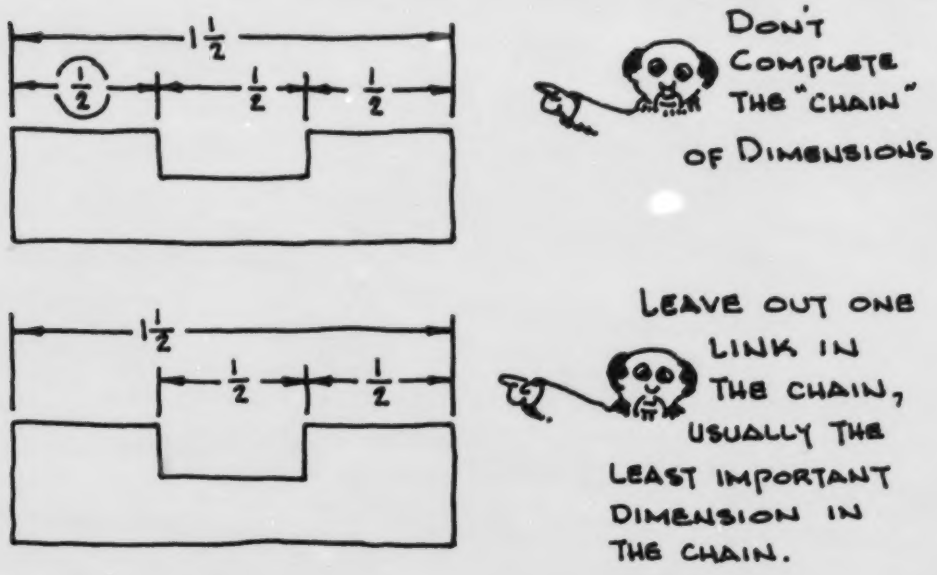


FIG. 4-10

If you were to complete the chain, you would be guilty of using "superfluous" dimensions. A superfluous dimension is one that isn't really needed, but is given anyway. Avoid giving too much information or unnecessary dimensions. This can be done by using a little common sense, notes, and practice. Take time to double-check your work so you won't give too much *or* too little information.

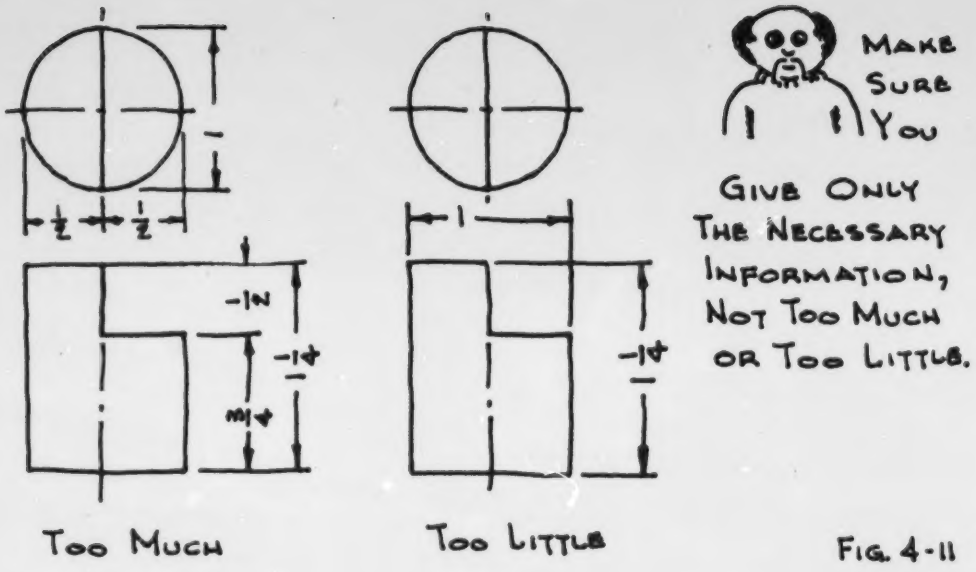


FIG. 4-11

FIG. 4-12

BASELINE

Baseline or datum-line dimensioning is when you use a point, surface, or center line as a reference to locate your dimensions. When used as a reference, the point becomes a base or datum point, the surface becomes a base or datum surface, and the center line becomes a base or datum line. All dimensions given will *start at a datum line, datum point, or datum surface,* but they have to start from the same datum line, point, or surface. The rectangular, polar, and chordal dimensioning systems all use datum-line dimensioning (see Unit III).

The sequence of operations is important in this system, since it will affect the choice of datum lines, points, and surfaces. *Datums must be stable.* If you are planning to use a point, line, or surface to measure from, it has to be there when the time comes to make the measurement. *You can't dimension from something that doesn't exist!*

For this reason, the first datum is the first machine operation to be performed. Notice that we are using the first *machine operation,* rather than manufacturing

operation. This is because you could start with a casting, a piece of rolled stock, or just about anything else, but what you want to know is "What do I do now?"

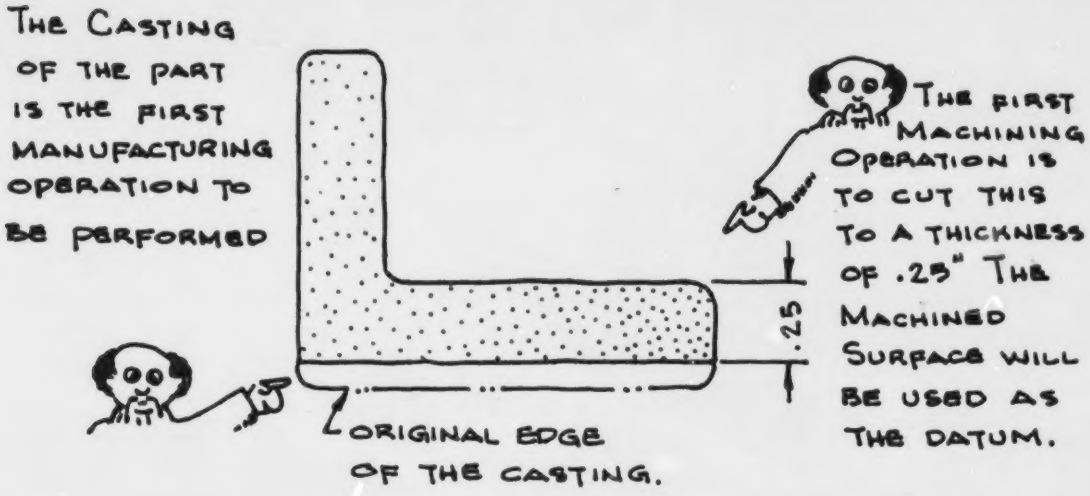

THE CASTING OF THE PART IS THE FIRST MANUFACTURING OPERATION TO BE PERFORMED
THE FIRST MACHINING OPERATION IS TO CUT THIS TO A THICKNESS OF .25" THE MACHINED SURFACE WILL BE USED AS THE DATUM.
.25
ORIGINAL EDGE OF THE CASTING.

FIG. 4-13

When you dimension using the datum-line system, you would *start from the datum* and mark off the first dimension, *then return to the datum* and mark off the second dimension. This would continue until the view is completely dimensioned *or* until it is necessary to change the datum (because of the sequence of operations). When you change from one datum to another, you will have to locate the new datum with respect to the old one (how far is it from the old datum, etc.).

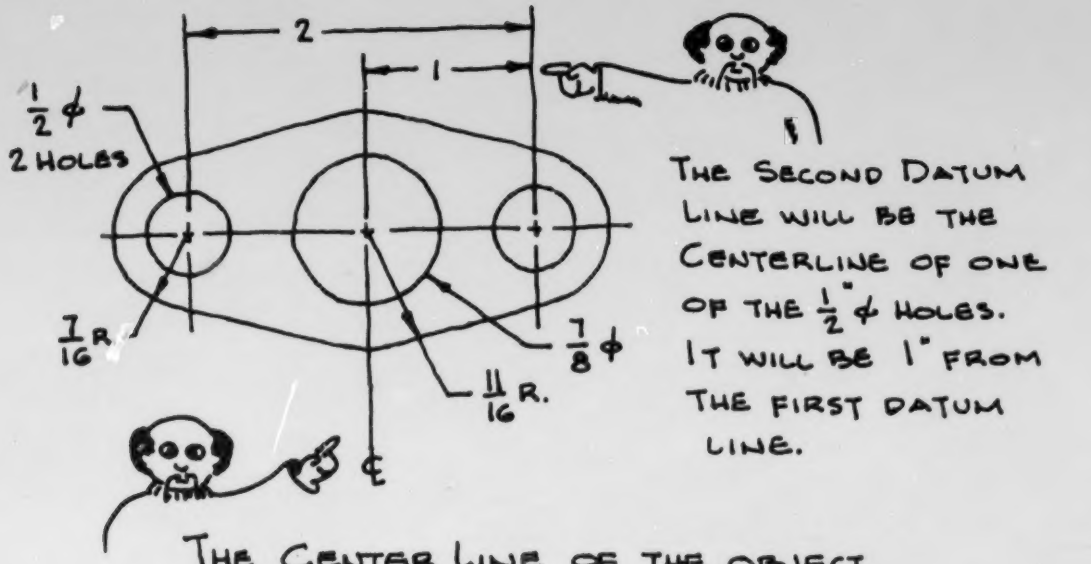

2
1
$\frac{1}{2}\phi$
2 HOLES
$\frac{7}{16}$R
$\frac{7}{8}\phi$
$\frac{11}{16}$R.
THE SECOND DATUM LINE WILL BE THE CENTERLINE OF ONE OF THE $\frac{1}{2}$" ϕ HOLES. IT WILL BE 1" FROM THE FIRST DATUM LINE.
THE CENTER LINE OF THE OBJECT WILL BE THE FIRST DATUM LINE (ALSO CALLED A BASELINE.)

FIG. 4-14

You will have a minimum of two datums per view, and these will probably be used in other views as well. *Be sure of what you are using as a datum, so it can be located in another view.* The datums *always* will be perpendicular or parallel to each other.

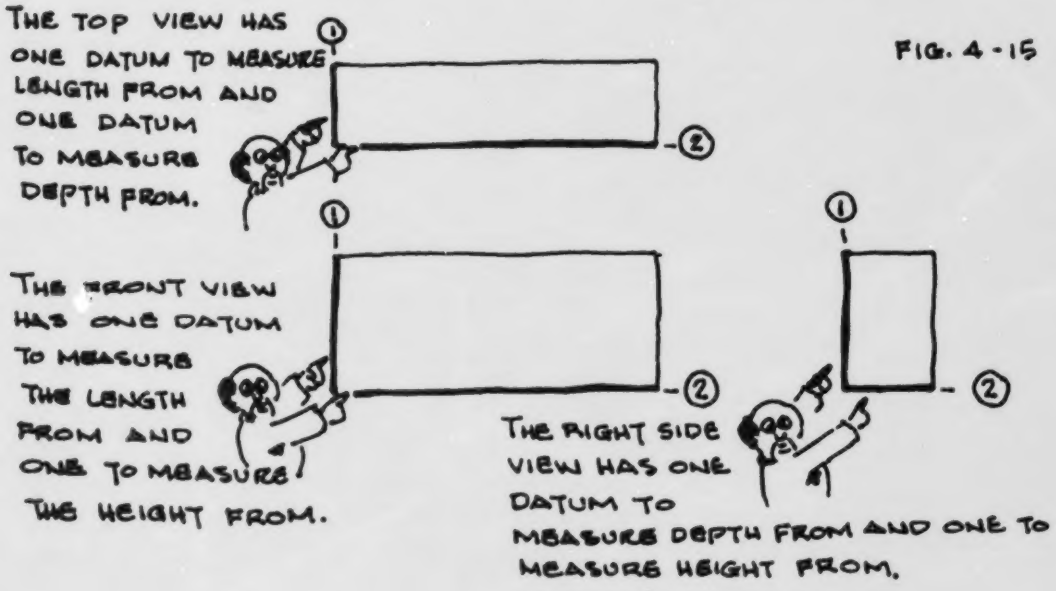
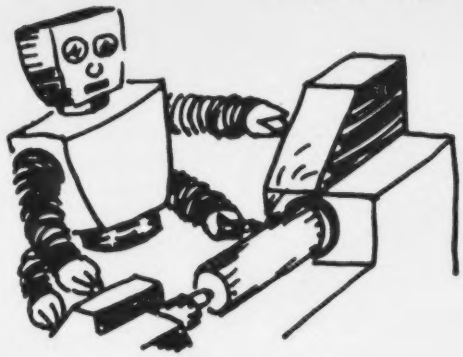

THE TOP VIEW HAS ONE DATUM TO MEASURE LENGTH FROM AND ONE DATUM TO MEASURE DEPTH FROM.
THE FRONT VIEW HAS ONE DATUM TO MEASURE THE LENGTH FROM AND ONE TO MEASURE THE HEIGHT FROM.
THE RIGHT SIDE VIEW HAS ONE DATUM TO MEASURE DEPTH FROM AND ONE TO MEASURE HEIGHT FROM.
1
2

FIG. 4-15

There will be times when your drawings will look as if they were chain-dimensioned, even though you used datum-line dimensioning. This is because the datums aren't marked or easily recognizable. In simple datum-line dimensioning, they are never marked. There are more complex forms of datum-line dimensioning which require the marking and identification of all datums -- numerical control or arrowless dimensioning and true-position dimensioning.

AUTOMATED MACHINIST?!

FIG. 4-16



Numerical Control (N.C.) machine tools have taken over in many industries today. These computer-operated machines save wasted hours and wasted material. They can perform tasks as complex as drilling 100 holes or making 40 spot welds at the same time. *But,* the computer is only as good as the information it is supplied, and *you* are supplying the information.

To aid you in supplying information the computer can understand (computers have limited vocabularies), arrowless dimensioning was developed. Arrowless dimensioning combines datum-line dimensioning, rectangular coordinates (called Cartesian coordinates), and tabular dimensions (sometimes). This system uses three datum lines, all perpendicular to each other at the same point and marked X, Y, Z. The X datum (also called axis) is horizontal, the Y datum is vertical, and the Z datum is also horizontal but is 90° to both the X and Y axes.

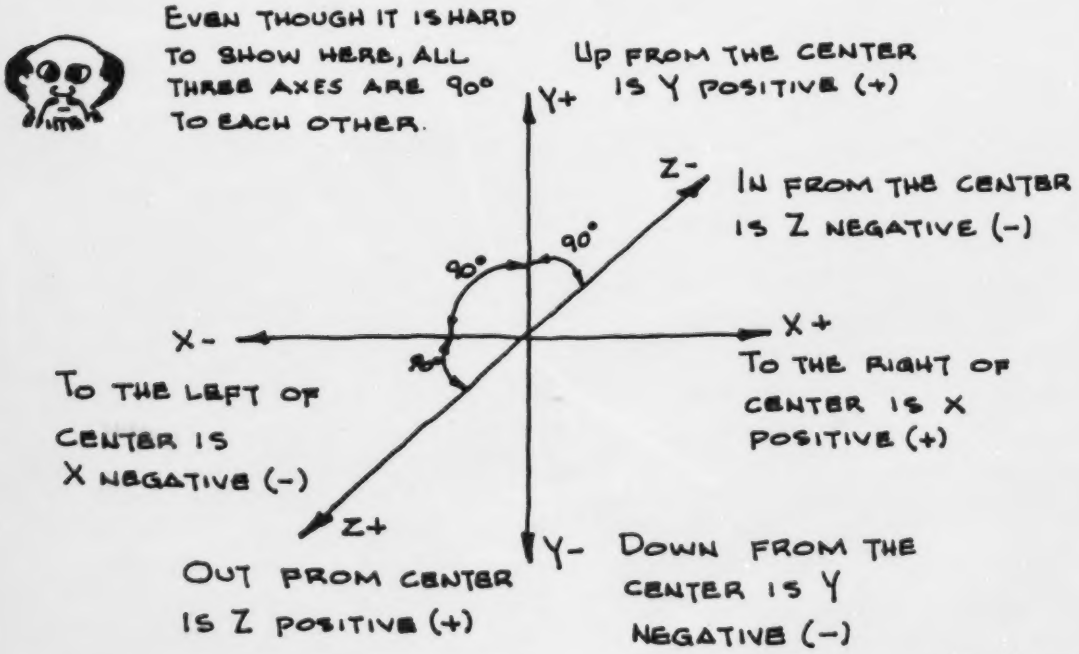

EVEN THOUGH IT IS HARD TO SHOW HERE, ALL THREE AXES ARE 90° TO EACH OTHER.
UP FROM THE CENTER IS Y POSITIVE (+)
Y+
Z-
IN FROM THE CENTER IS Z NEGATIVE (-)
90°
90°
90°
X-
X+
TO THE LEFT OF CENTER IS X NEGATIVE (-)
TO THE RIGHT OF CENTER IS X POSITIVE (+)
Z+
Y-
OUT FROM CENTER IS Z POSITIVE (+)
DOWN FROM THE CENTER IS Y NEGATIVE (-)

FIG. 4-17

Since the computer wouldn't understand you if you told it to go right or left, move in or out, move up or down, a simpler way of giving instructions had to be devised (remember, computers aren't that smart). Minus and plus are used to control the computer's movement along any datum line:

1. X+ = Any horizontal movement along the X axis to the right of where the three datum lines cross (the origin).
2. X- = Any horizontal movement along the X axis to the left of the origin.
3. Y+ = Any vertical motion along the Y axis, above the origin.
4. Y- = Any vertical motion along the Y axis, below the origin.
5. Z+ = Any horizontal motion along the Z axis, away from the origin.
6. Z- = Any horizontal motion along the Z axis, in from the origin.

The object or workpiece to be dimensioned is placed so that a square or rectangular object will have only the positive (+) coordinate values for X and Y, and only the negative coordinate values for Z when dimensioned. This would mean the bottom, left, front corner would be the origin.

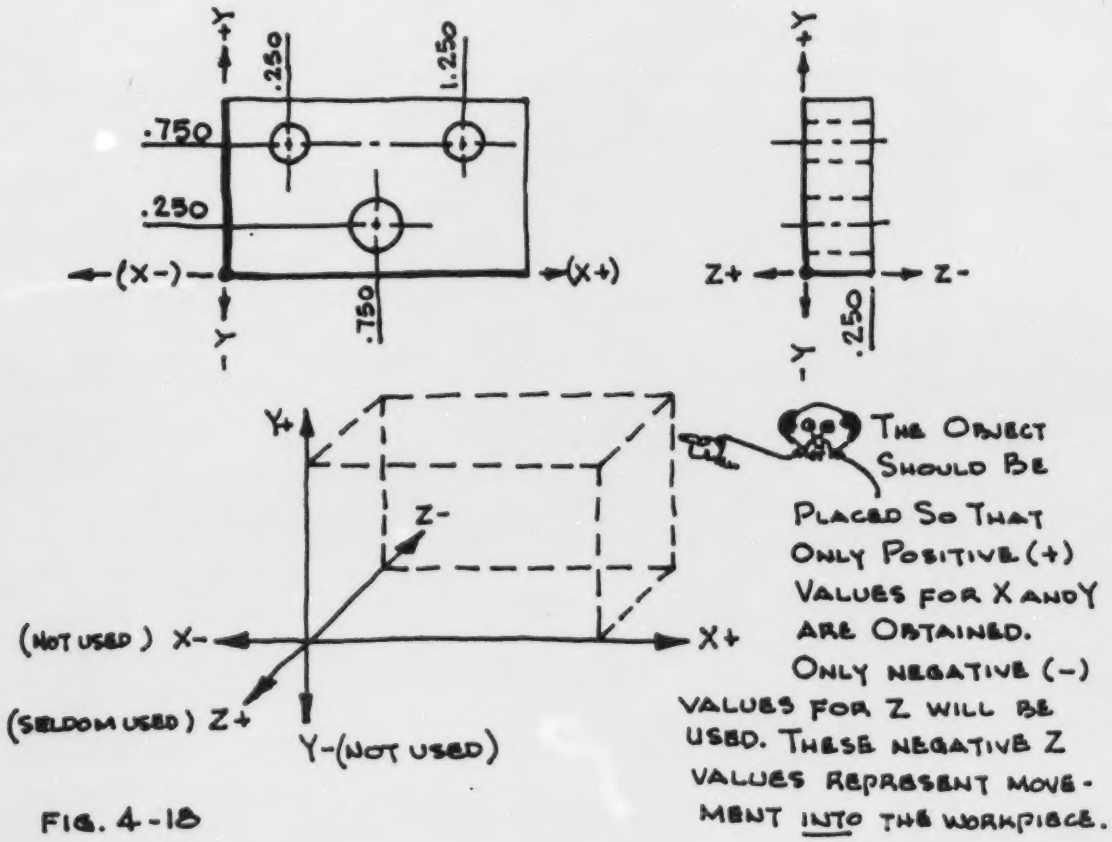


FIG. 4-18

A circular, cylindrical, or spherical object is positioned so that the centerlines of the object correspond (match) with the X and Y datum lines.

This means you would have both positive (+) and negative (–) values for X and Y when dimensioning the object.

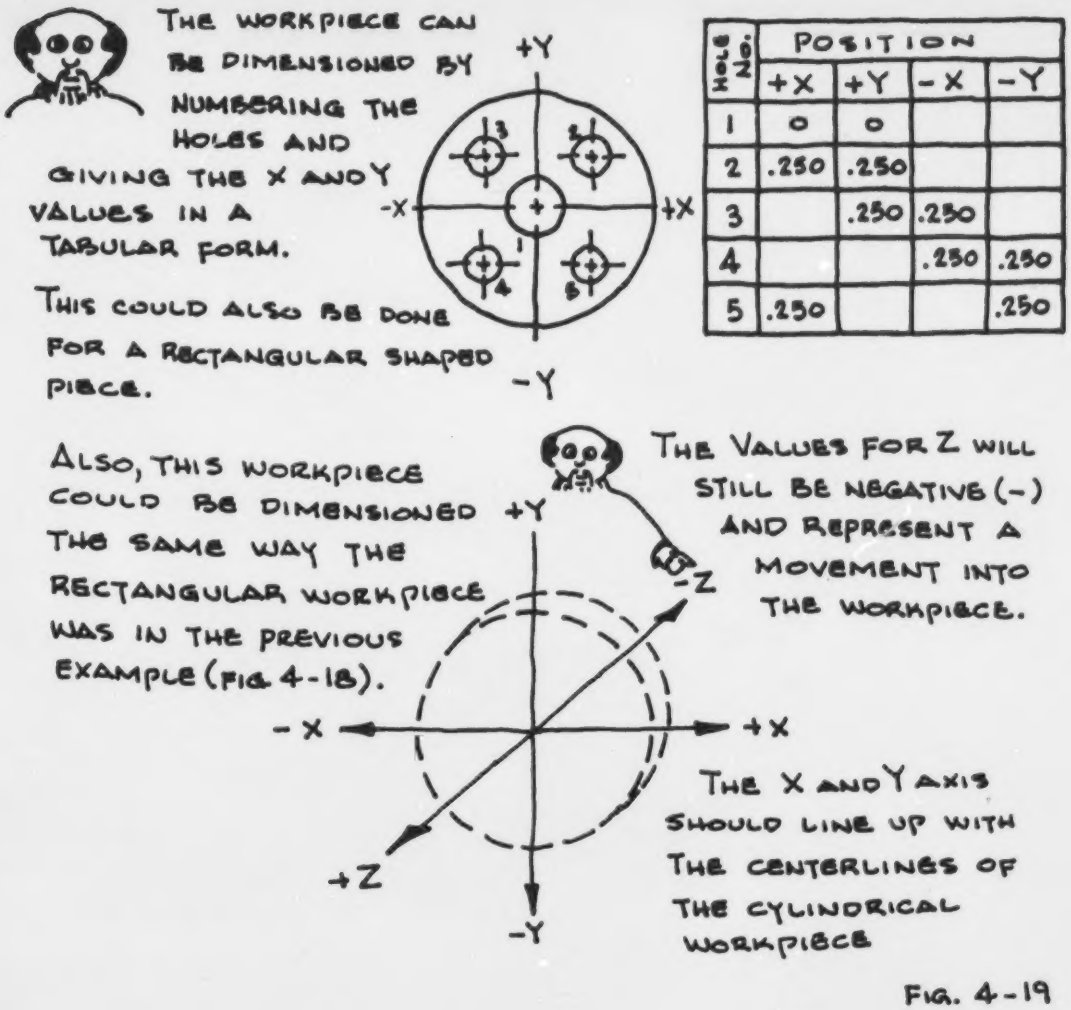


FIG. 4-19

Notice that Figure 4-18 has the dimensions just above the extension lines and Fig. 4-19 has the dimensions listed in tabular form. Both are correct. Also, notice that when the dimensions are placed above the extension lines, the dimensions increase as you go up or to the right.

This would be the same as if you had dimension lines and were measuring from the datum line.

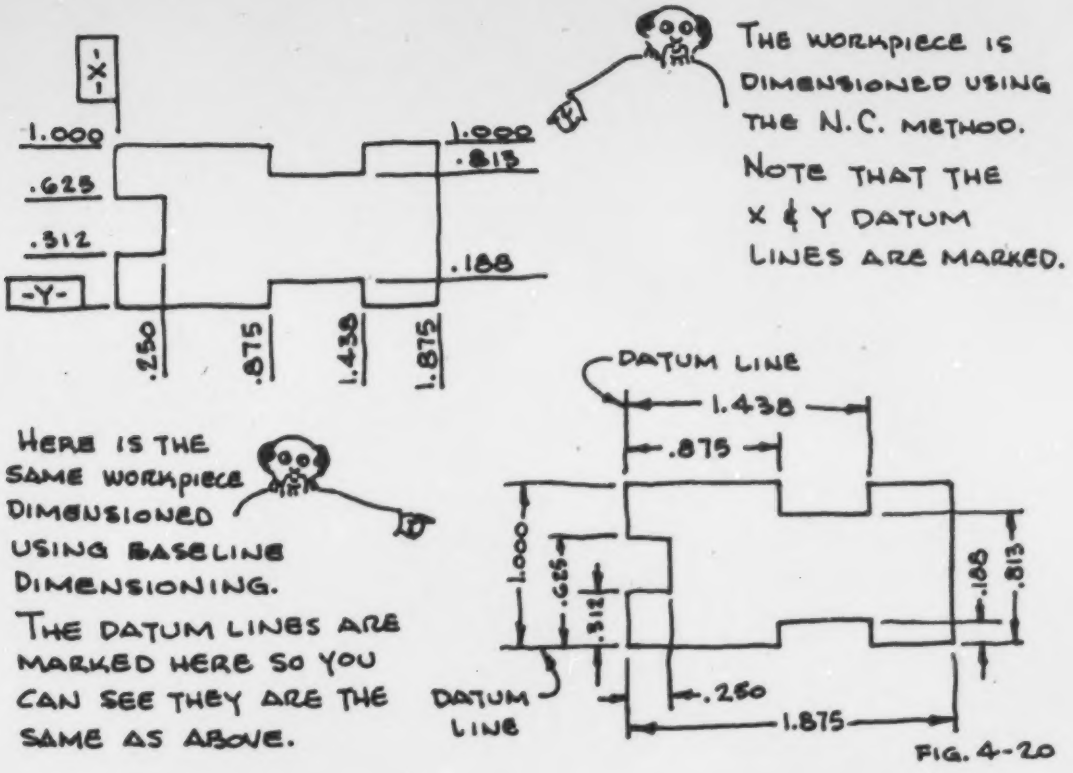


FIG. 4-20

TRUE POSITION

True-position dimensioning uses the best parts of datum-line and numerical-control dimensioning. The datums are marked, like numerical control drawings, but they *don't* have to meet at one point. (What do you call the point where the X, Y and Z datums cross?) Dimensions are given the same as datum-line dimensions are. By combining the two systems, the information can be used by a machinist or a computer.

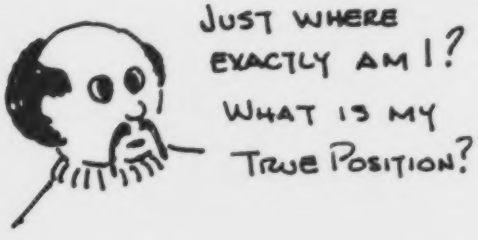


FIG4-21

The marked datums are based on a few major machine operations, and all other operations are called out in relation to those operations. Also, the datums are marked alphabetically -- A, B, C, etc. -- rather than X, Y, and Z. Since the datums don't have to cross at a given point, it isn't necessary to use a grid system or negative dimensions.

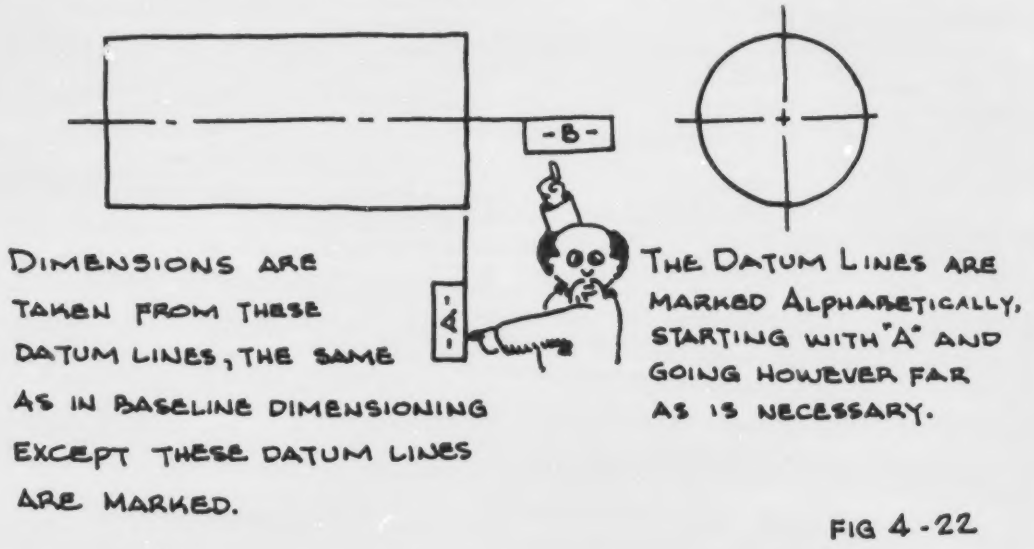

-B-
-A-
DIMENSIONS ARE TAKEN FROM THESE DATUM LINES, THE SAME AS IN BASELINE DIMENSIONING EXCEPT THESE DATUM LINES ARE MARKED.
THE DATUM LINES ARE MARKED ALPHABETICALLY, STARTING WITH "A" AND GOING HOWEVER FAR AS IS NECESSARY.

FIG 4-22

Also, it is not necessary to take all your dimensions from a marked datum. So, you will be working with marked and unmarked datums.

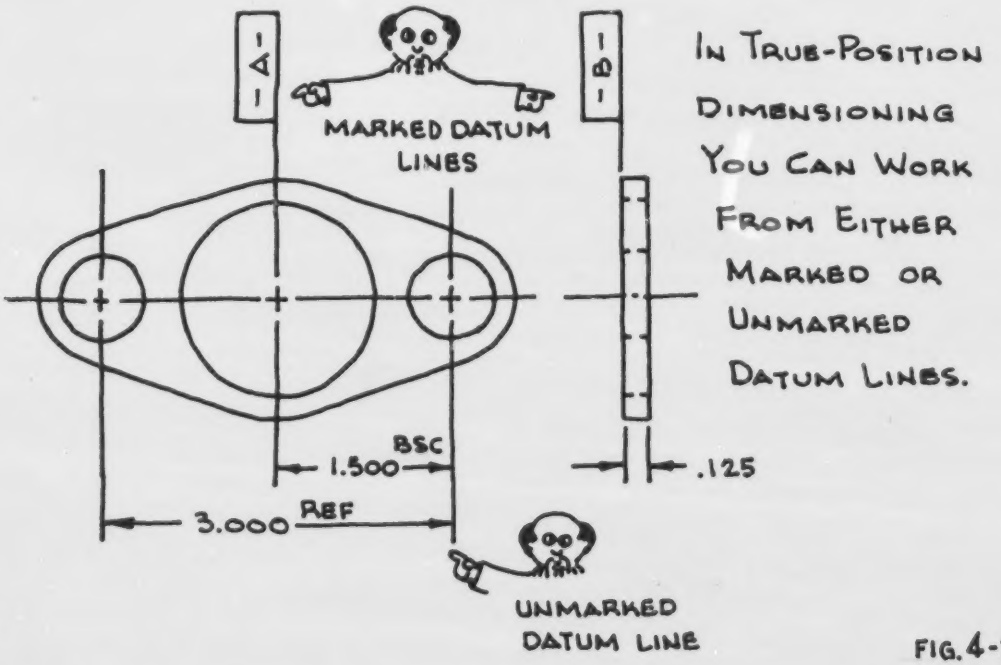


FIG. 4-23

The practice unique to true-position dimensioning is its use of *basic* and *reference* dimensions. Reference dimensions are used to check other dimensions and are marked with the letters "REF". For example, if two holes are to each be 1.500" from a central hole, they would be 3.000" from each other. The 3.000" dimension would be a reference dimension used to check the distance between the two outside holes.

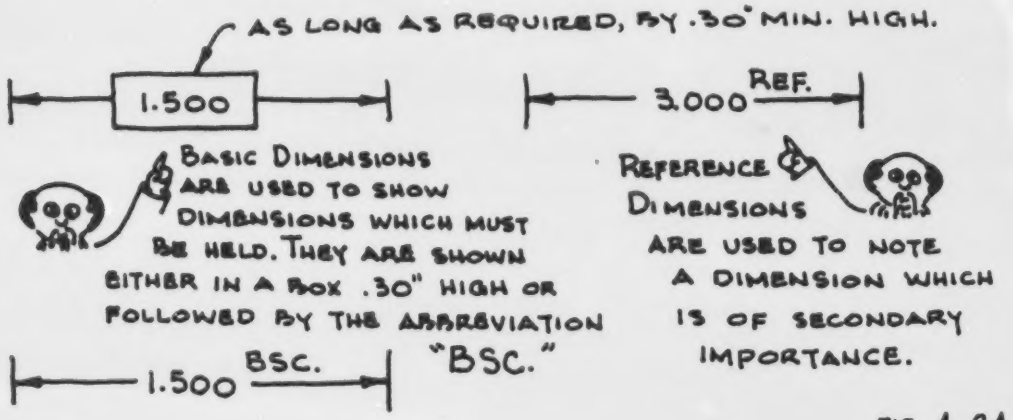

AS LONG AS REQUIRED, BY .30" MIN. HIGH.
1.500
3.000 REF.
BASIC DIMENSIONS ARE USED TO SHOW DIMENSIONS WHICH MUST BE HELD. THEY ARE SHOWN EITHER IN A BOX .30" HIGH OR FOLLOWED BY THE ABBREVIATION "BSC."
REFERENCE DIMENSIONS ARE USED TO NOTE A DIMENSION WHICH IS OF SECONDARY IMPORTANCE.
1.500 BSC.

FIG. 4·24

The basic dimensions are "theoretical values used to describe the exact size, shape, or position of a feature".[4] This means the dimensions are "what should be" — the size it ought to be, if it were perfectly made. Since no object can be made perfectly, the basic or true-position dimensions are only references to check how far from the theoretical dimension the actual measurements are. The dimensions are marked by the word "basic", the abbreviation "BSC", or a rectangular box around the dimension.

---

[4] *Engineering Drawing and Graphic Technology,* 11th Edition; French, Thomas E. and Vierck, Charles J.; McGraw–Hill Book Company; Copyright 1972, pp. 376–377.

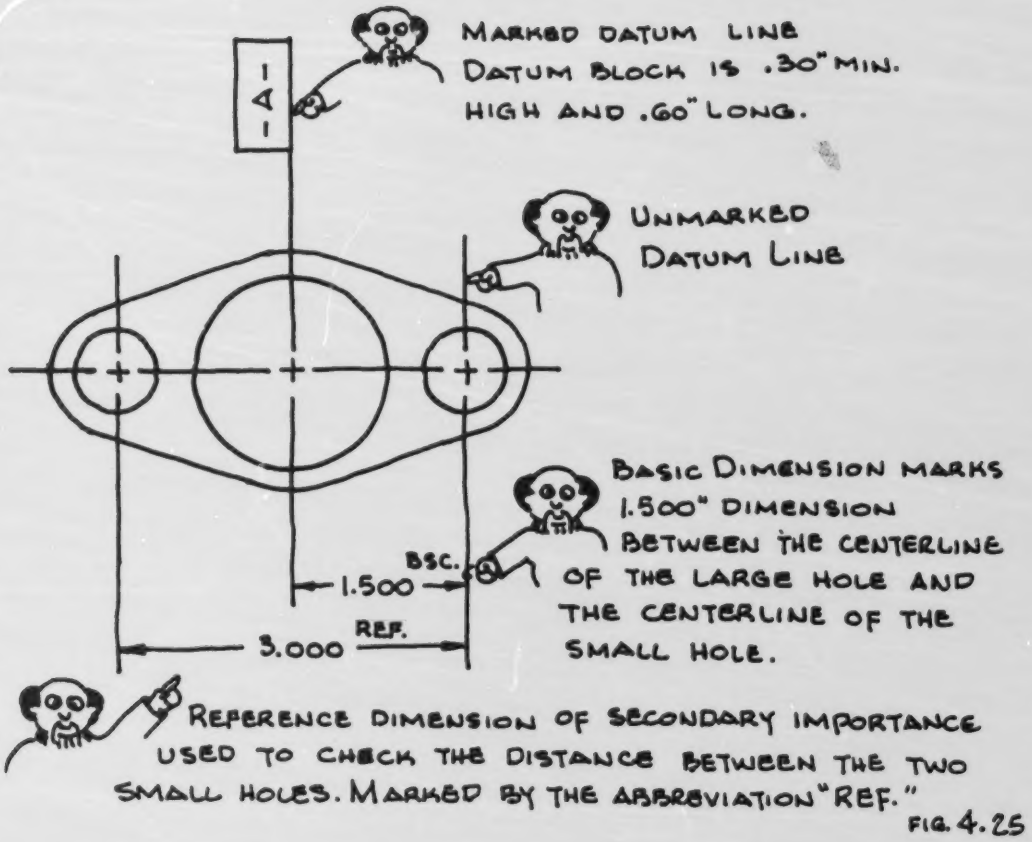


FIG. 4.25

Other characteristics of true-position dimensioning will be discussed in the unit on Geometric Tolerancing. This is done because geometric tolerancing and true-position dimensioning were developed to be used together.

THINK ABOUT IT!

Why do you think it is necessary to standardize things like dimensioning systems and the way they read?

Can you think of why dimensions are read from the bottom or the right-hand side of a drawing? Think about the logic involved in the decision to use these standards.

Chain dimensioning has its limitations, but can you understand why it was originally used? Think about the advantages it has for the draftsman.

Baseline or datum-line dimensioning has advantages over chain dimensioning, and if you can realize what they are, it might be easier to use and understand.

Why do you think variations of baseline dimensioning were developed? (i.e., Numerical Control and True-Position Dimensioning.)

With technology advancing as rapidly as it is, think about the changes that are taking place in the drafting profession. Can you name some of the changes taking place?

EXERCISE YOUR BRAIN

What two "patterns" are used to read dimensions? Give their names and explain what the names mean as well.

What are the advantages and disadvantages of each of the above dimensioning "patterns"?

What effect do manufacturing methods and sequence of operations have on how the dimensions are placed on the drawing and which dimensions are used?

Explain chain dimensioning, its advantages and disadvantages.

Explain simple baseline or datum - line dimensioning, its advantages and disadvantages.

How was the datum-line dimensioning system altered to meet the needs of today's growing technology?

Explain the methods used in dimensioning drawings for use with computerized machining methods.

Explain the Cartesian Coordinate System and how it is used to give directions to computers. How many datums are there and how are they marked?

Explain how true-position dimensioning and numerical-control dimensioning are alike and how they are different.

Explain what the terms "reference" and "basic" dimensions mean and how they are marked on a drawing.

UNIT V: PRODUCTION PROCESSES

CHOICE

Production processes include all manufacturing methods used by industry. Obviously, you won't be able to dimension or specify all of them on your drawings. The majority of these methods will be specified by the engineer, designer, and/or production manager, and you will have to use them on your drawings. The production manager and the machinist choose the actual methods of making an object and the order they will be performed, based on the engineer's and designer's suggestions.

If you're wondering, "Why should I worry about learning production processes when other people are going to tell me what to do anyway? If I need to know anything, I can ask." You can ask if you need to know something, but you can't make yourself a pain in the neck.

FIG.5.1

The people who make the decisions have the responsibility to do just that, and are paid accordingly. You have the responsibility to take those decisions and use them in your drawings. As far as the company is concerned, their time is worth more than yours. It might sound harsh, but industry is based on economics -- the greatest profit for your money. When you take up the engineer's, designer's, or production manager's time unnecessarily, you are cutting the company's profits.

This doesn't mean you shouldn't ask any questions. When you start a job, you are expected to ask questions -- a lot of them -- because you are learning. However, you are also expected to understand what is being explained to you and quickly. If you have already learned part of it in school , you have a head start. Work is not like school; when you don't learn on the job, *you get fired.*

The information presented in this unit will help you understand what the engineers, designers, production managers, and machinists are talking about, and help you to put this information on your drawings.

CASTING

Casting is a method of forming molten metal by the use of a mold, which has the shape of the object to be made. The mold can be made of sand (sand casting), plaster (plaster-mold casting), ceramics, or metal (die or investment casting). The casting -- the finished product -- is rough and usually requires further machining.

FIG 5-2

Sand casting uses a wooden pattern, made slightly larger than the size of the finished casting. The wooden pattern is placed in a box of tightly packed sand. This forms the lower half of the mold, called the *drag*. Then more sand is packed around the pattern until it is completely covered. This is called the *cope*. Two holes are also left in the cope; one for pouring in the metal (sprue) with a larger area at the top (pouring basin) and one to allow the air, gas, and excess metal to escape (riser). When the pattern is removed, the shape is left in the sand, and the metal will take that shape when its poured into the mold.

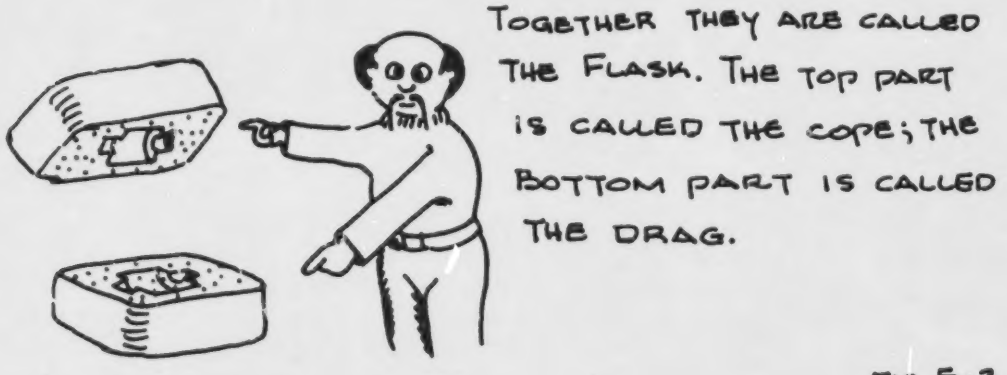


FIG. 5-3

The wooden pattern is larger than the finished casting because the metal cools and shrinks. The amount it will shrink can be predicted and measured. When this measurement has been made, it is *added* to the size of the casting to give the pattern size. Also, all corners are rounded on the pattern, since it is very hard to obtain a sharp corner on a sand casting. The rounded corners are called fillets and rounds. The dimensioning of the fillets and rounds is taken care of in a general note.

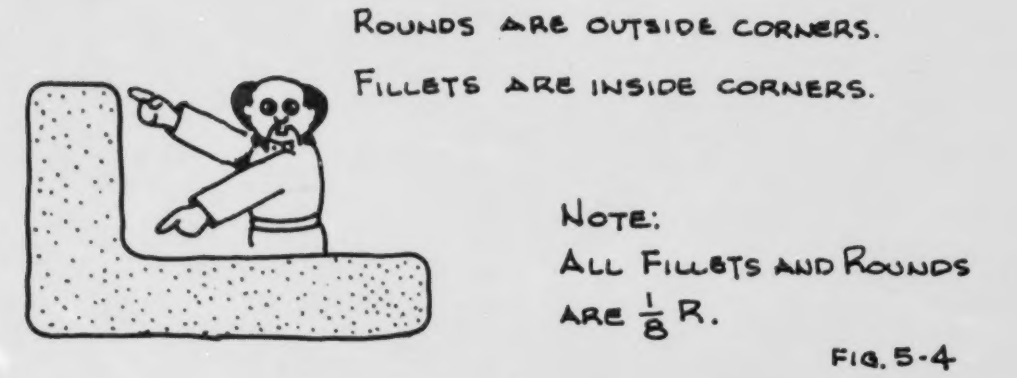


FIG. 5-4

When the metal shrinks away from a sharp corner, it causes a fault — a crack or hollow spot — in the casting. The fault can cause the casting to break up during the machining process and make it useless. The corner can also break away or be imperfectly formed. Both problems can cause casting failure and cost the company a great deal of money in wasted materials.

When the object being cast has a hole in it, a *core* is used. A core is made of baked sand and has the shape of the *inside* of the object to be cast. A *core box* is made of wood and has the desired shape cut into it. The sand is packed into the core box and then baked so it will keep its shape. The core is then placed in the mold so that when the casting is poured it will have the desired shape.

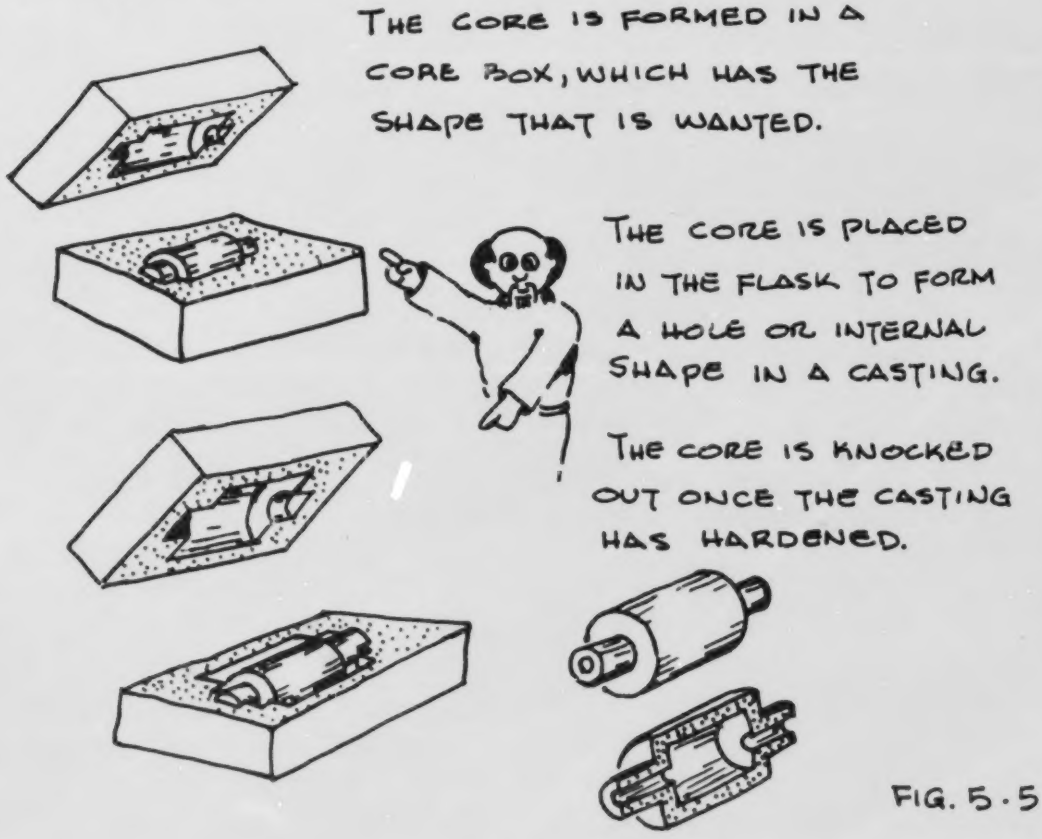

THE CORE IS FORMED IN A CORE BOX, WHICH HAS THE SHAPE THAT IS WANTED.
THE CORE IS PLACED IN THE FLASK TO FORM A HOLE OR INTERNAL SHAPE IN A CASTING.
THE CORE IS KNOCKED OUT ONCE THE CASTING HAS HARDENED.

FIG. 5-5

Die-casting is similar to sand casting. The mold (called a die) is made of metal instead of sand, and one part of the mold is removable, so you can get the casting out. Also, the metal is forced into the mold, using a plunger or ram. The final casting is of better quality than the sand casting and can be used to make more castings than the sand mold. Sharp corners can be obtained with this method.

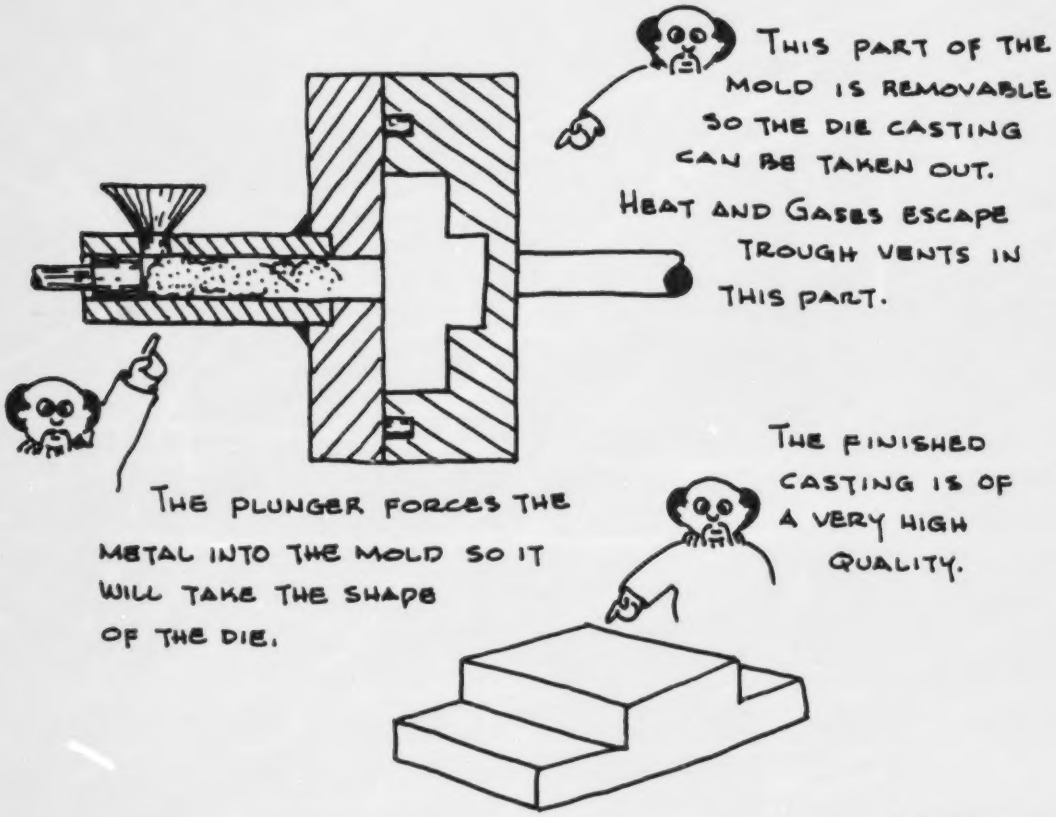

THIS PART OF THE MOLD IS REMOVABLE SO THE DIE CASTING CAN BE TAKEN OUT.
HEAT AND GASES ESCAPE TROUGH VENTS IN THIS PART.
THE PLUNGER FORCES THE METAL INTO THE MOLD SO IT WILL TAKE THE SHAPE OF THE DIE.
THE FINISHED CASTING IS OF A VERY HIGH QUALITY.

FIG.5-6

Investment casting uses wax or plastic to make a disposable pattern. A *flask* or *shell* is formed *over* the pattern. In the flask method, a metal flask is placed over the pattern and filled with a material that will turn hard when left in the air (plaster or ceramic materials). In the shell method, the pattern is dipped into a liquid ceramic until a thick coat has been formed.

In both cases, the flask and shell methods, the molds are baked after they have hardened. When heated, the wax or plastic melts and flows *out* of the mold, leaving its shape in the mold. Metal is then poured into the mold to make the final casting. The mold is spun or vibrated during the casting process to make sure the metal gets into all the corners. Sharp corners can be obtained through investment casting, as well as a high quality casting.

Other forms of castings are based on these basic types of casting: sand casting, die-casting, and investment casting. For more information on the basic types of casting and special types of casting, check your library for books on foundry practices and manufacturing methods.

ROLLING

Rolling is another way of producing stock shapes and sizes. In rolling, the metal is forced between two or more cylindrical rollers -- like a kitchen rolling pin. The rollers can have a plain, flat surface, like a rolling pin, or they can have a shape (grooves) cut into them, like a candle stick or a spindle. Also, the rollers are turning in opposite directions, so that once the metal has been started between the rollers, it will continue to be drawn into the rollers.

FIG 5·7

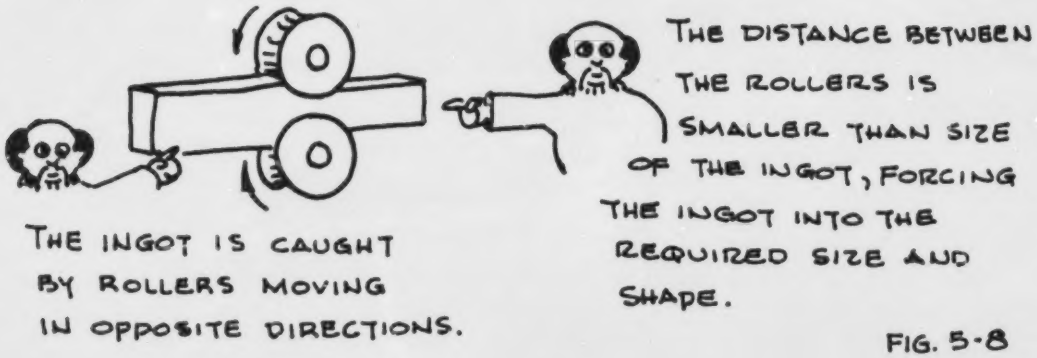

THE INGOT IS CAUGHT BY ROLLERS MOVING IN OPPOSITE DIRECTIONS.
THE DISTANCE BETWEEN THE ROLLERS IS SMALLER THAN SIZE OF THE INGOT, FORCING THE INGOT INTO THE REQUIRED SIZE AND SHAPE.

FIG. 5·8

The metal can be either "hot" or "cold" when it is rolled. Hot-rolled metal is heated to a temperature between "red" and "white" hot, but *not to its melting point.* The metal will be "plastic" at this stage -- soft and pliable -- needing less roller pressure to form than cold metal. Railroad tracks, I-beams, and other shapes with a fairly rough surface are formed by hot-rolling.

Cold-rolled metal has been hot-rolled into a general shape, cleaned in an acid solution, and then dipped in a lime solution to neutralize the acid. This process cools the metal below the "red hot" stage, but the metal is still not cool or cold. It is harder than the hot metal,though.

Since the metal is harder, the rollers must put more pressure on the metal to get it to take a special shape or form. Each time the metal is run through the rollers, it gets harder and gets a smoother surface. This hardness with a smooth surface is the advantage of cold-rolling.

FORGING

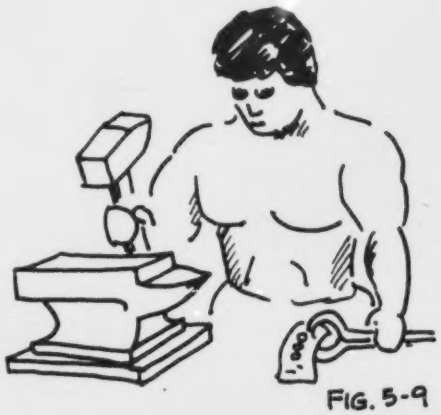


FIG. 5-9

Forging is the method of shaping hot metal, *not molten,* by hammering or squeezing it into a form. A blacksmith forges objects on an anvil -- horseshoes, swords, and maybe checks. Besides blacksmithing, there is drop-forging and press-forging.

Drop-forging uses a mechanical hammer -- pneumatic or steam -- to hit the hot metal and force it into a *die* or *pattern.* The mechanical hammer has more power than any blacksmith, so larger pieces can be made. Also, since metal dies are used, the parts can be made again and each will be the same.

Press-forging doesn't use hammering to form the hot metal. Instead, the metal is *squeezed* into shape. The dies are in two parts, half the shape cut into one die and half into the other. The hot metal block is placed between the two dies and then the press is closed very slowly until the two dies meet. The metal is forced into the die openings, creating the desired shape.

Remember, forging differs from die-casting since the metal is only heated -- not molten. Also, forged parts are stronger than cast *or* machined parts. The reason for this is that metal has a "grain", like wood, once it has been rolled. Since forged parts are made from hot rolled ingots, the grain is already formed. When the metal is forced into the die, nothing is done to break or cut the grain pattern -- the hot metal "stretches" to keep grain unbroken.

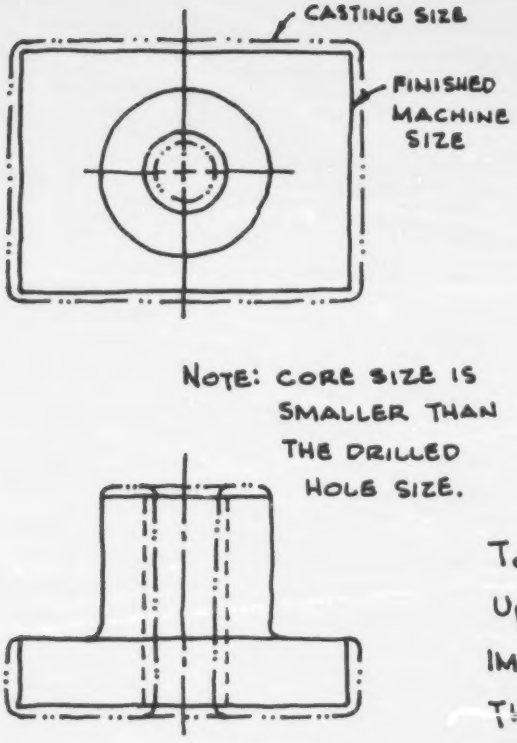
Most stock metal is rolled, having its own grain. When it is cut, the pattern is broken; this causes a weakening of a metal. Though this weakening is not great enough to affect most parts, where strength is of importance, it will make a difference.

Castings don't have a grain pattern like rolled or forged parts. Instead, they have a coarser grain that is more easily broken or fractured. Castings are like soft woods, soft and more easily split; while rolled stock and forgings are like hard woods, closer grained and tougher.

FIG. 5-10

All of the *forming* methods talked about so far require only a general note to indicate which method of forming is to be used and the material (steel, cast iron, etc.) to be used. Only castings may require special drawings.

The reason for this is the cast part will be one size when it is cast and another after it has been machined. There is no problem when one drawing shows the cast or wooden pattern size (pattern drawing), and another shows the machined size (machine drawing); but when they are combined on one drawing, you may have some problems. When this is necessary, the outline of the unmachined casting is shown in *phantom lines*, while the shape of the machined casting is shown by object lines, as it normally would be. Only extremely important areas, like cored holes, are dimensioned on the pattern part of the drawing. The machined casting is dimensioned as usual.



CAST SIZES ARE ALWAYS LARGER THAN THE PLANNED MACHINED SIZES, AND THE CORE SIZES ARE ALWAYS SMALLER THAN THE DRILLED HOLE SIZES — TO ALLOW FOR SQUARING UP AND CUTTING AWAY IMPERFECTIONS IN THE CASTING.

FIG. 5-11

MACHINING

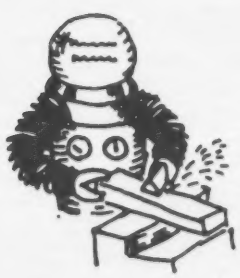


FIG. 5-12

Machining means any method of shaping metal through the cutting or chipping of metal. This cutting or chipping can be done to a rough casting or a piece of rolled stock. It can also be done in many ways: filing, grinding, sawing, drilling, etc. The basic machine processes and those most commonly used by industry will be explained. Also, the methods of dimensioning each process will be explained.

DRILLING

The making of holes is important to industry, since they are used to connect parts, locate mating parts, locate other processes, and other things. The most common way of machining a hole is drilling.

FIG 5-13

Drilling is one machine process that is easily recognizable by most people. How many people really understand how a drill works is another question. The drill is turned or spun at high speed, and the point cuts into the surface of the metal. The sharp cutting edge on the point is called the lip, and each drill has at least two lips -- but they don't talk or kiss. Also, the spiral grooves, called flutes, carry the metal up to the surface, keeping the drill from jamming and breaking off.

THE LIP IS THE EDGE WHICH CUTS OR CHIPS THE METAL AWAY WHEN THE DRILL BIT IS TURNED AT HIGH SPEED. THE FLUTES CARRY THE METAL UP AWAY FROM THE LIP SO THE DRILL BIT WON'T JAM AND BREAK IN THE HOLE BEING DRILLED.

FIG. 5-14

The point is also used to locate and start the drilling process. When the machinist begins the drilling process, he first locates where the hole is to be drilled by drawing two intersecting (crossing) and perpendicular lines -- the same as you will do on your

drawing. At the point where the two centerlines cross, the machinist uses a punch to indent the surface. This is to make sure the drill point is lined up properly and won't slip from that location.

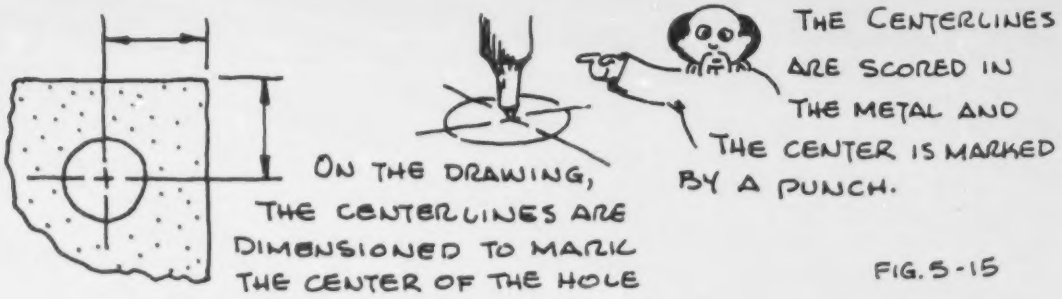

THE CENTERLINES ARE SCORED IN THE METAL AND THE CENTER IS MARKED BY A PUNCH.
ON THE DRAWING, THE CENTERLINES ARE DIMENSIONED TO MARK THE CENTER OF THE HOLE

FIG. 5-15

Also, the point must be shown on your drawing when you aren't looking straight into the hole and when the hole is *not* drilled through the metal. The point angle (see drill diagram) is actually between 120° and 135°, but you will use the 120° angle on your drawings.

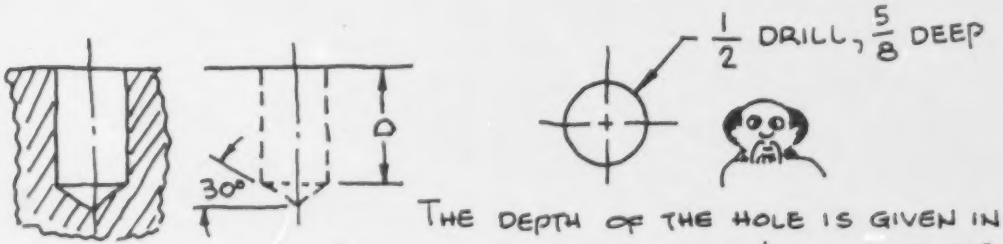


FIG. 5-16

Drill sizes are given in millimeters, inches (fractions and decimals), letters, and whole numbers. In the English measurement system, the letters (A to Z) and whole numbers (#80 to#1) are used to fill in the gaps between the standard fractional drill sizes, and each has a decimal equivalent (i.e., #1 drill = .2280 and an "A" drill = .2340).

When dimensioning a drilled hole, you will give the drill size and the depth of drilled hole in a local note, after you have located the two centerlines. Lettered and whole-number drill sizes require the decimal equivalent to be given as well (see appendix

or textbook). The letter "A" is the smallest lettered drill size and the letter "Z" is the largest.The number 80 drill is the smallest numbered drill size and the number 1 the largest. The depth of the drilled hole is measured from the surface to the beginning of the point, *but not including* the point. This dimension can be given in the local note or on the drawing.

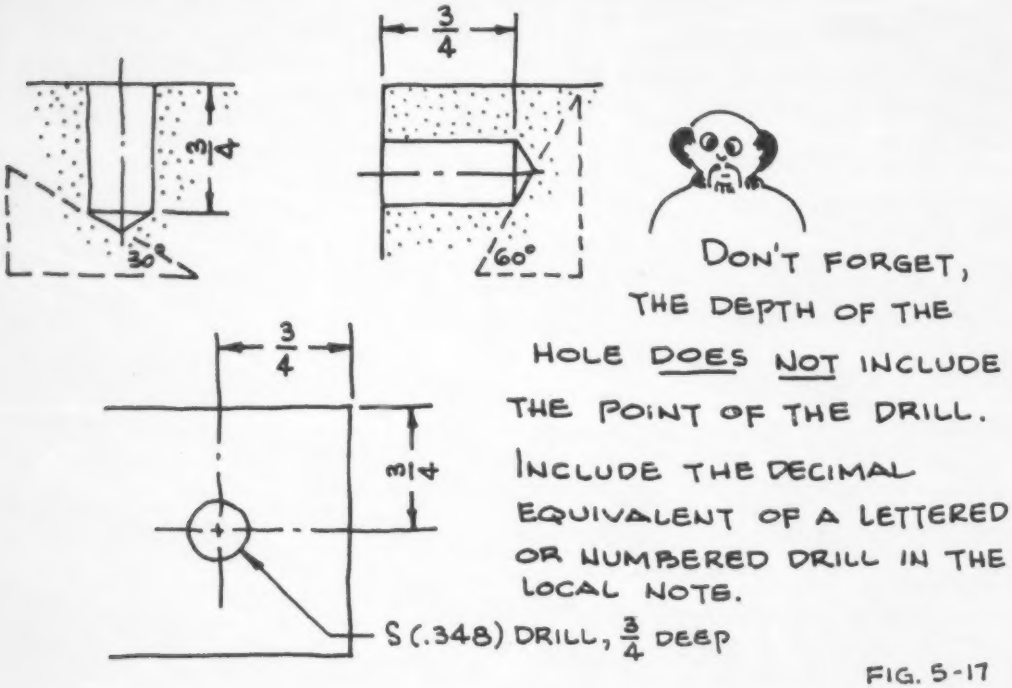

3/4
MIN
30°
3/4
60°
MIN
DON'T FORGET, THE DEPTH OF THE HOLE DOES NOT INCLUDE THE POINT OF THE DRILL.
INCLUDE THE DECIMAL EQUIVALENT OF A LETTERED OR NUMBERED DRILL IN THE LOCAL NOTE.
S (.348) DRILL, 3/4 DEEP

FIG. 5-17

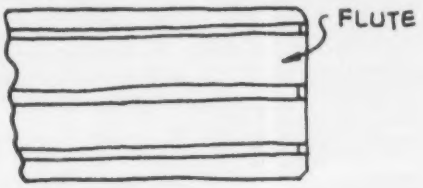
REAMER

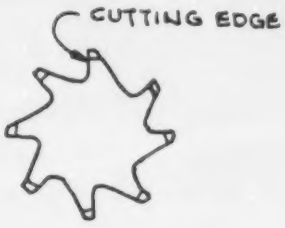
Reaming is similar to drilling, but it gives a smoother surface and is used to enlarge a hole. The reamer has flutes, like the drill, that are spiral - shaped or straight. The complete edge of the reamer flute cuts,

FIG. 5-18

instead of the point of the drill. The reamer has no point, so the flutes are the only things that can cut. This also means there must be a hole to begin with, called a pilot hole.

THE REAMER HAS STRAIGHT FLUTES INSTEAD OF SPIRAL FLUTES LIKE THE DRILL.

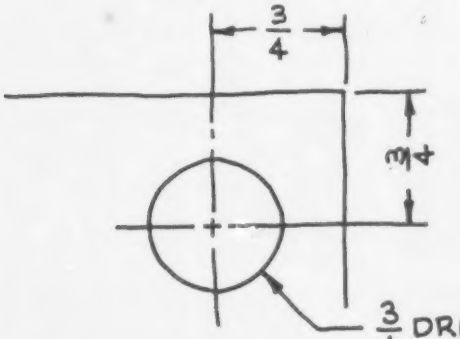




SINCE THE REAMER DOESN'T HAVE A POINT TO START THE CUTTING, THERE MUST BE A HOLE NEARLY THE SAME SIZE AS THE REAMER. THIS IS CALLED A PILOT HOLE.

FIG 5-19

When dimensioning a reamed hole, it is again necessary to locate its center lines first. The diameter of the reamed hole and its depth are then given in a local note. When the size of the pilot hole is known,it is placed before the reamed size in the local note.



WHEN DIMENSIONING A REAMED HOLE, GIVE THE DRILLED HOLE SIZE FIRST AND THEN GIVE THE REAMED HOLE SIZE. THIS IS BECAUSE THE OPERATIONS WILL BE DONE IN THAT ORDER.

FIG. 5-20

Reamed holes are used to allow clearance for a shaft, bolt or stud; to make tapered holes (larger at one end than the other) for taper pins; to obtain accurate hole sizes for drive, spring, lock, and dowel pins.

BROACHING

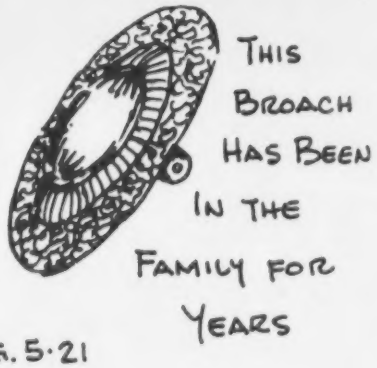


FIG. 5·21

Broaching is a method of machining holes other than round holes. The broaching tool is forced, pushed, or pulled through a drilled hole. The tool teeth begin at a size slightly larger than the pilot hole and continue to get larger until the final size is reached. The action of the tool is both cutting and chipping, leaving a very smooth finish on the hole.

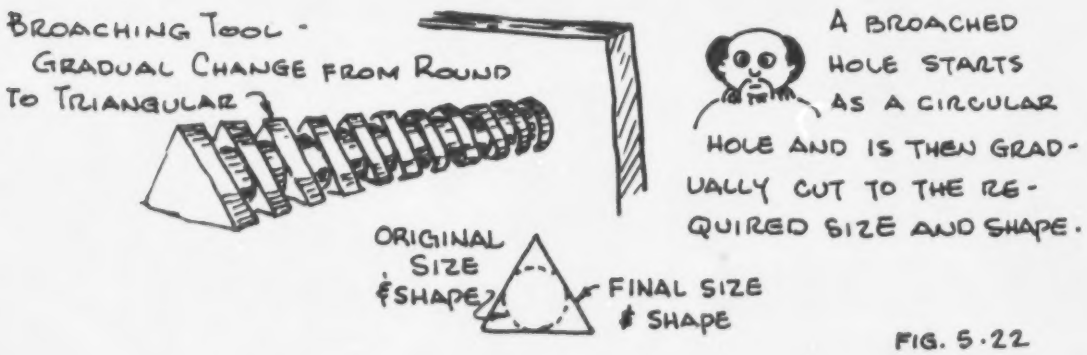

BROACHING TOOL - GRADUAL CHANGE FROM ROUND TO TRIANGULAR
ORIGINAL SIZE & SHAPE
FINAL SIZE & SHAPE
A BROACHED HOLE STARTS AS A CIRCULAR HOLE AND IS THEN GRADUALLY CUT TO THE REQUIRED SIZE AND SHAPE.

FIG. 5·22

Dimensioning a broached hole requires only the location of its center by two centerlines, and dimensioning the size and shape in a local note. It is not necessary to note the pilot-hole size on your drawing, unless it is required by the company you work for, or unless you are making a pattern drawing for a part that will be broached later.

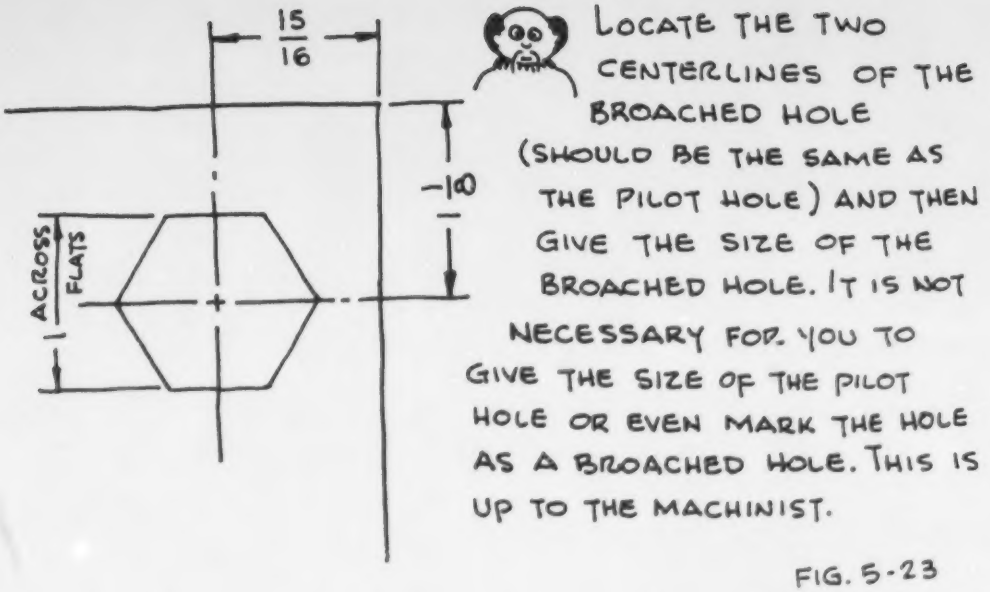

15/16
ACROSS FLATS
LOCATE THE TWO CENTERLINES OF THE BROACHED HOLE (SHOULD BE THE SAME AS THE PILOT HOLE) AND THEN GIVE THE SIZE OF THE BROACHED HOLE. IT IS NOT NECESSARY FOR YOU TO GIVE THE SIZE OF THE PILOT HOLE OR EVEN MARK THE HOLE AS A BROACHED HOLE. THIS IS UP TO THE MACHINIST.

FIG. 5-23

Broached holes are used in engine blocks, gears, rifles, cannons, and many other products. Key-ways are cut using broaching tools, as well as splines and other necessary connection methods.

BORED

Boring uses one or more replaceable cutting blades to enlarge and/or give a smoother surface to a hole. It sounds as if boring and reaming are almost the same thing, except for the tool that's used. *This is wrong* — they are not the same.

FIG 5-24

First, the boring process can be performed on a wide variety of equipment: lathe, milling machine, drill press, or boring machine. The more equipment that a process can be performed on, the less money it costs — normally. Second, the size of the hole can be varied using the boring tool. This means a number of operations can be performed using one cutting tool, also making the

processes cheaper. Third, boring is not limited to standard sizes due to the cutting tool. You can bore holes to any size that's required, making it more versatile than reaming.

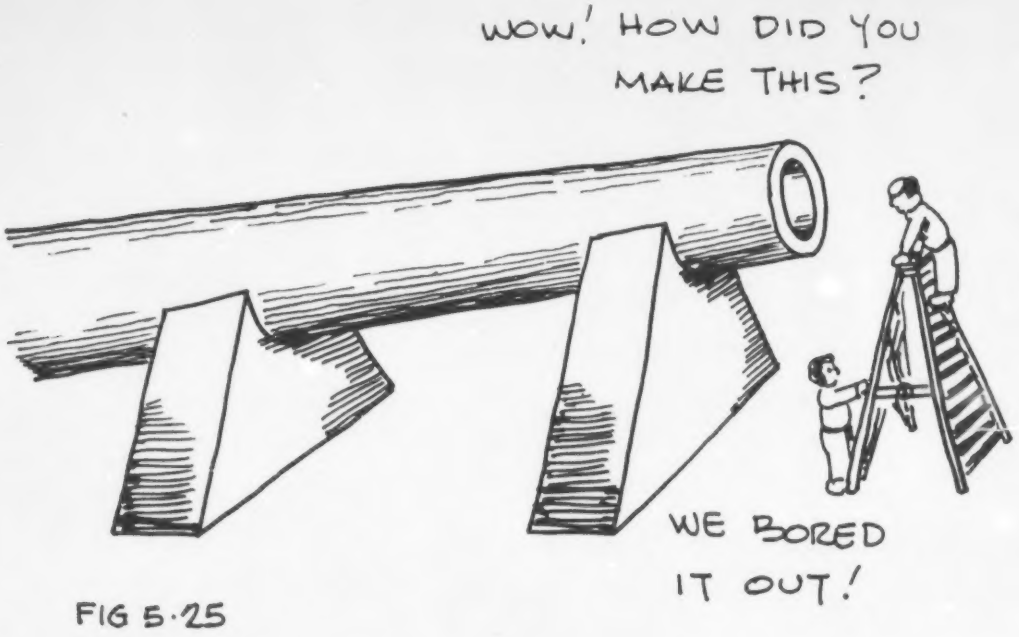


FIG 5·25

When you dimension a bored hole, it is necessary to locate the center lines and give the diameter of the bored hole in a local note. If a pilot hole is drilled, then its diameter must also be given in the local note.

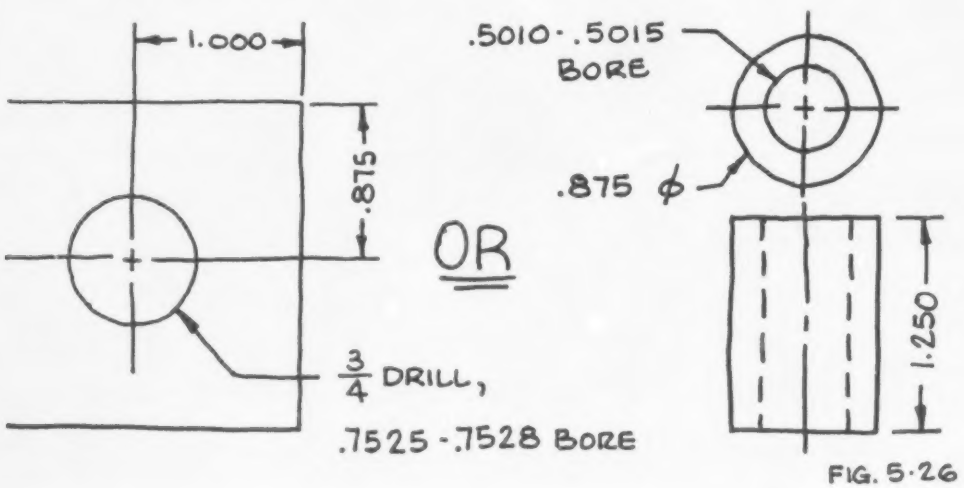


FIG. 5·26

COUNTERBORE & COUNTERSINK

A counterbore is a large hole made above a smaller hole. The sides of the two holes are parallel, and both holes have the same center lines. The small hole is usually drilled but can be reamed or bored. The large hole is always bored.

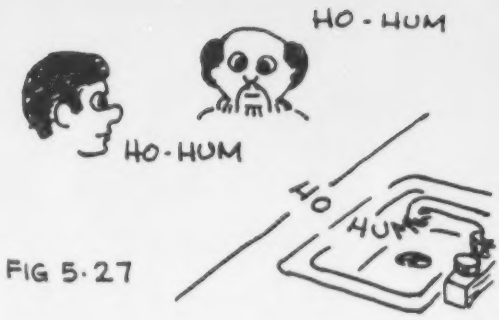


FIG 5·27

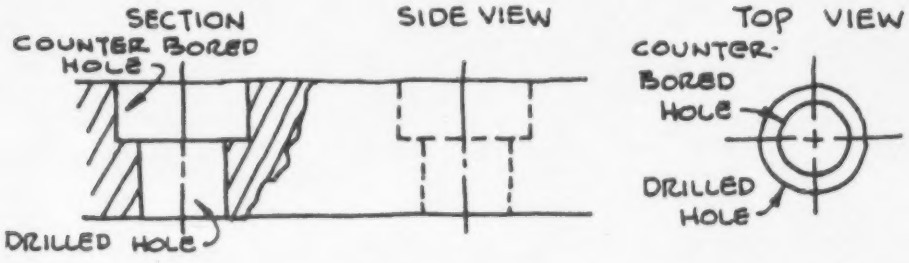


FIG. 5·28

There are three ways to make a counterbore: (1) by the use of a boring tool; (2) by the use of a counterbore tool; or (3) by the use of a combination drill. The counterbore is normally used to make the head of a socket-head machine screw flush with the surface of an object. Since there are standard size machine screws, standard counterbore tools are also made just for this purpose.

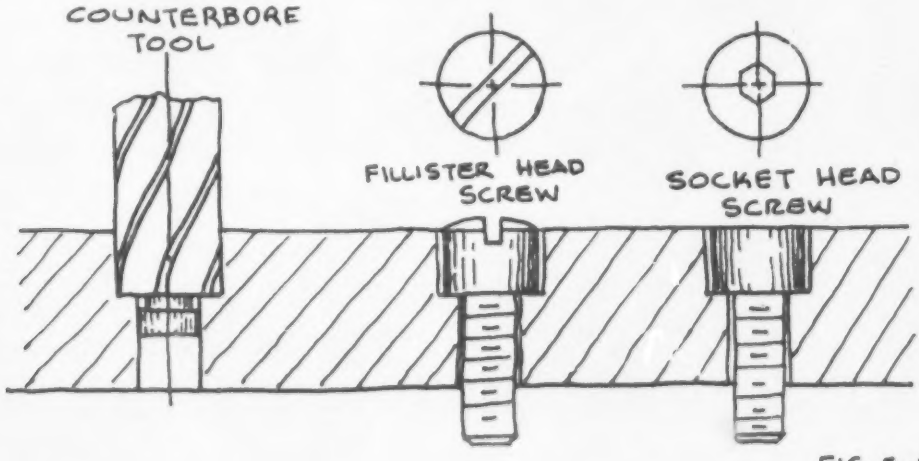


FIG. 5·29

When dimensioning a counterbore, you would locate the centerlines, like any hole, and use a local note to give the size and method of making the small hole, as well as the size and depth of the counterbore (c'bore). Don't worry about how the counterbore will be made; the machinist will choose the method -- you choose the size.

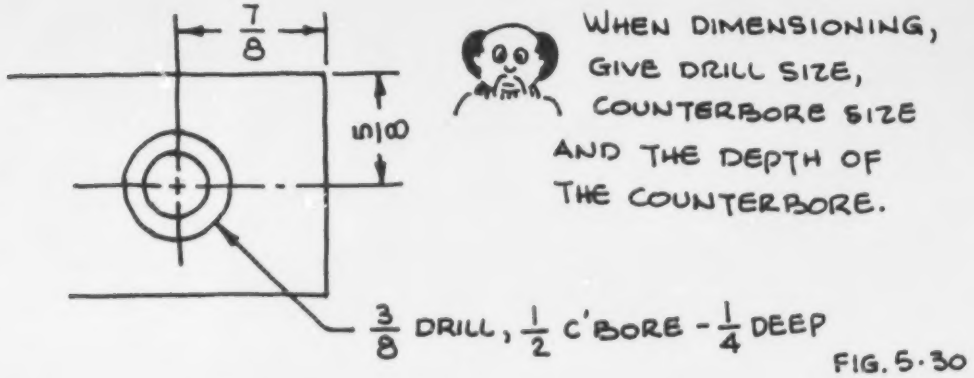


FIG. 5·30

A countersink is a method of changing from one diameter hole to a smaller diameter hole, smoothly. A countersink looks like the inside of a funnel; the sides slant down from the large diameter to the smaller diameter.

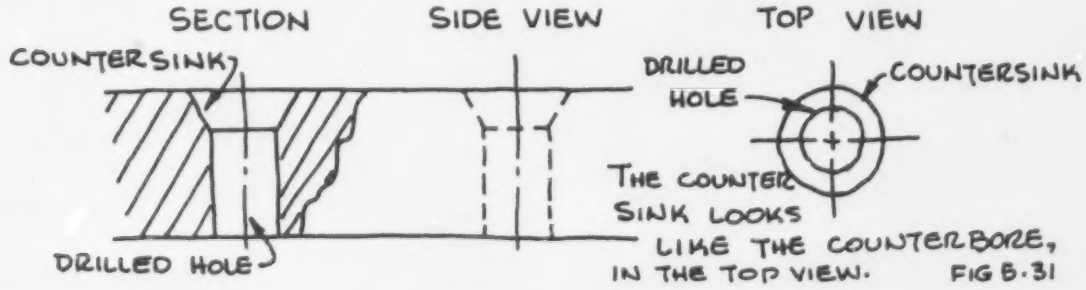


FIG 5·31

The small hole is drilled, reamed, or bored; then the countersink is made using a standard countersink tool or a boring tool. A method of making both the small hole and the countersink in the same operation is to use a combination drill. Countersinks are used primarily to make the head of a flat-head machine screw flush with the surface of an object. This is the reason for standard size countersink tools.

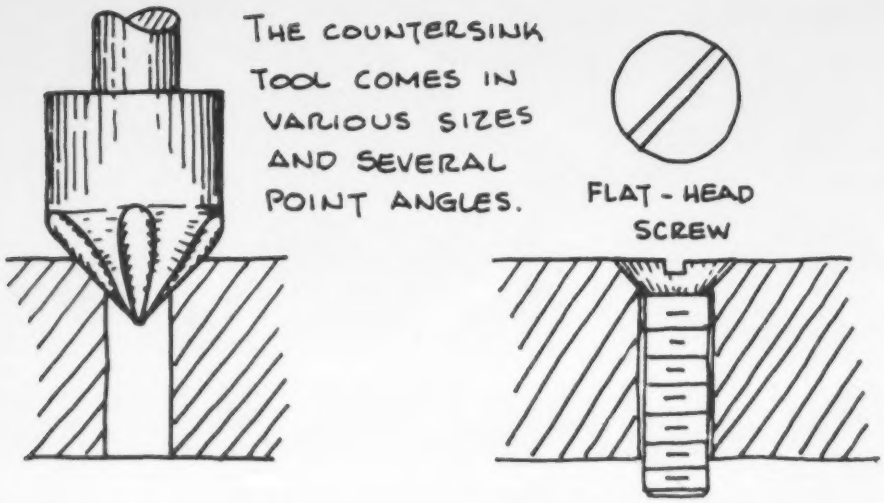


FIG. 5·32

Since the angle of the machine-screw head is standardized (60°, 80°, 82°, 90°, 100°, or 120°), you must include the angle of the countersink when you dimension it. Besides this, the process is almost the same as dimensioning a counterbore: locate the centerlines, give the diameter and method of making the small hole, and give the large diameter of the countersink (CSK). Instead of the depth, give the angle of the countersink.

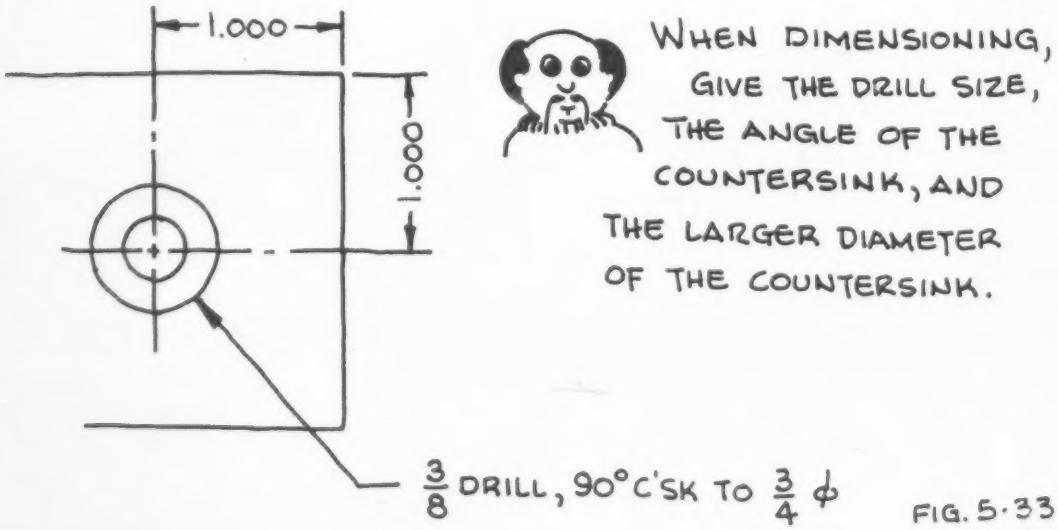

1.000
1.000
WHEN DIMENSIONING, GIVE THE DRILL SIZE, THE ANGLE OF THE COUNTERSINK, AND THE LARGER DIAMETER OF THE COUNTERSINK.
3/8 DRILL, 90° C'SK TO 3/4 ϕ

FIG. 5·33

SPOTFACE

There are two types of spotface: one is like a shallow counterbore and the other is a flat, circular area usually on top of a boss. In this case, the boss is not the guy who pays you, but a short cylindrical shape that sticks out from the surface of a cast workpiece.

FIG. 5·34

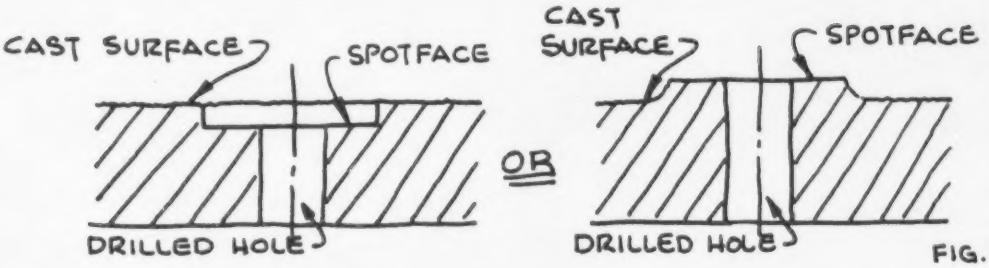
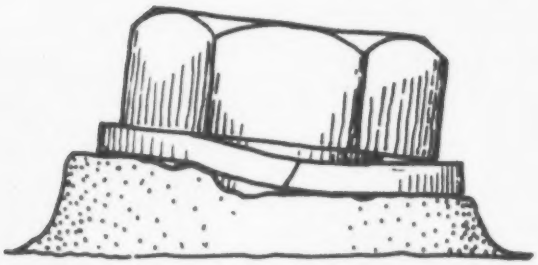


FIG. 5·35

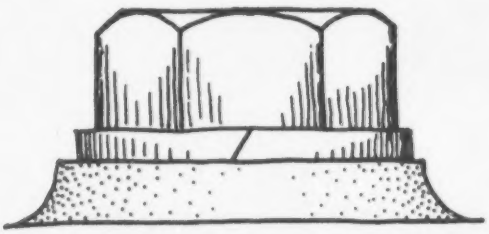
Both types of spotface are used to make a flat, smooth area around the top of a drilled, reamed, or bored hole, so the washer and/or bolt-head will sit flat on the surface of the cast or rough workpiece. This is done to prevent pressure from being put on only one part of the bolt-head, causing it to bind or shear (break off). Spotfaces are found almost entirely on castings, but can be found on other rough-surfaced objects.

ROUGH SURFACE



THE ROUGH SURFACE CAUSES THE BOLTHEAD TO SIT UNEVEN AND BREAK OFF.

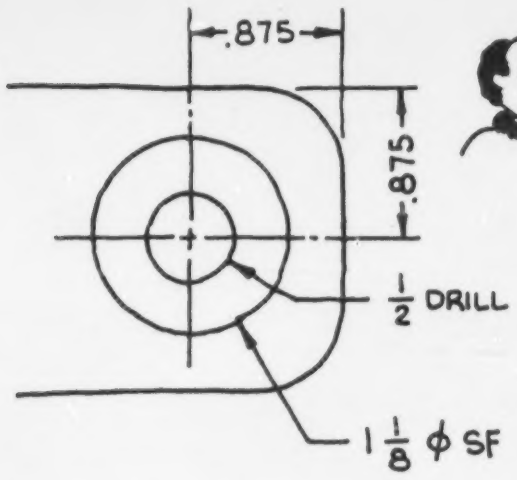
SPOTFACE



THE SPOTFACE ALLOWS THE BOLTHEAD TO SIT FLUSH WITH THE SURFACE.

FIG. 5·36

Dimensioning a spotface is taken care of in a local note. The local note contains the diameter of small hole, the method of making the hole (drilled, reamed, etc.), the diameter of the spotface, and the depth, when necessary. The centerlines must also be located.



THE SPOTFACE LOOKS LIKE THE COUNTERBORE OR COUNTERSINK IN THE TOP VIEW.

WHEN DIMENSIONING A SPOTFACE (SF), GIVE ITS DIAMETER AND DEPTH IF NECESSARY. THE DRILL SIZE IS GIVEN IN A SEPERATE LOCAL NOTE.

FIG. 5-37

LATHE WORK

The lathe is a machine that spins a piece of stock metal, and the metal is cut by a chisel-edge cutting tool. Very small amounts of metal can be cut from the stock -- as small as .0002". This means very precise parts can be made on the lathe. Also, the lathe can be used to perform a variety of operations. Remember, the more operations a machine can perform, the more money the manufacturer saves. The price of one machine that can perform several operations is less than the cost of several machines that can only perform one operation.

The operations which can be performed on the lathe are drilling, boring, reaming, facing, straight turning, taper turning, thread cutting, grooving, profiling, and knurling. It sounds like a lot and it is -- the lathe is one of the most versatile (all-purpose) machines found in the machine shop.

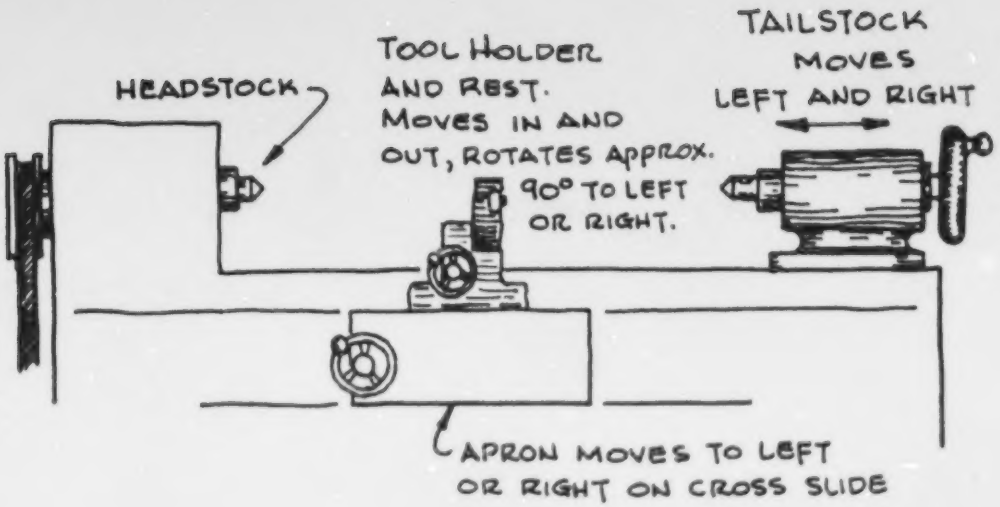

HEADSTOCK
TOOL HOLDER AND REST. MOVES IN AND OUT, ROTATES APPROX. 90° TO LEFT OR RIGHT.
TAILSTOCK MOVES LEFT AND RIGHT
APRON MOVES TO LEFT OR RIGHT ON CROSS SLIDE

FIG. 5-38

TURNING

Turning is a very simple operation, where the stock (the piece of metal or plastic being worked on) is spun at high speed and cut to a specified size. The finished shape will be cylindrical, regardless of its original shape.

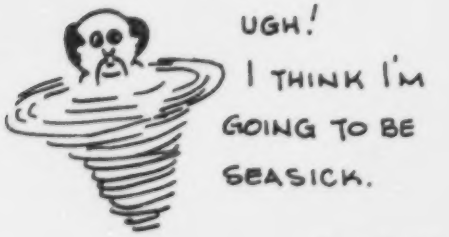


FIG. 5-39

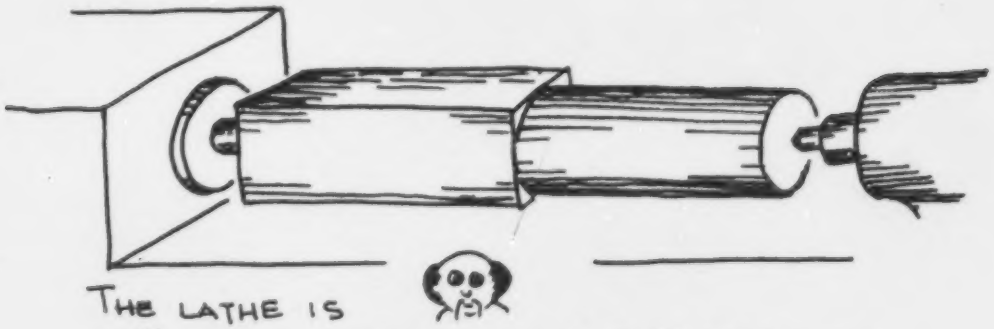


FIG. 5-40

In dimensioning straight-turning operations, the finished length and diameter of the object are all that is necessary. *No local note is used to call out the operation.* The

original stock size, shape, and material are given in the parts list. The parts list is located either in the upper right-hand corner of the drawing *or* directly above the title block.

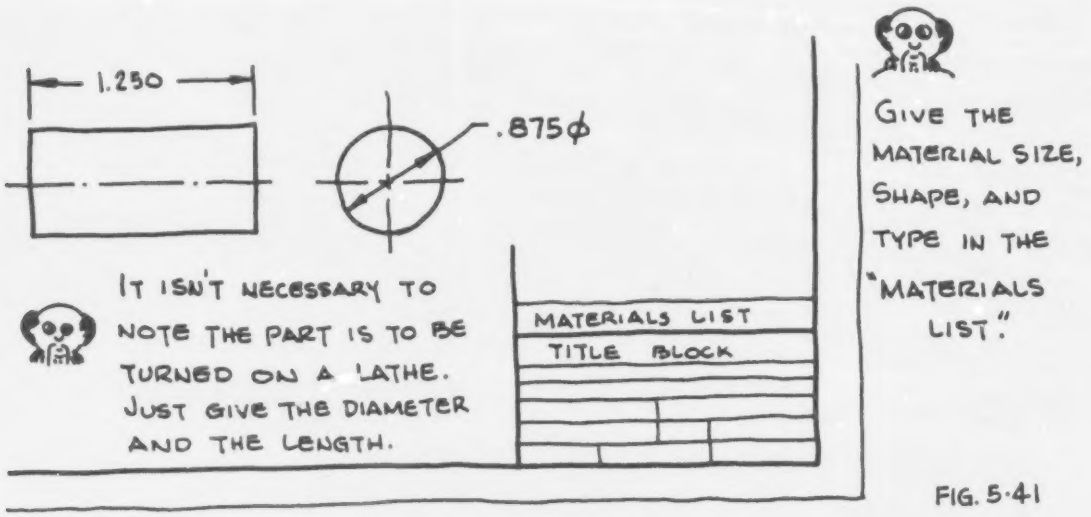

1.250
.875ϕ
IT ISN'T NECESSARY TO NOTE THE PART IS TO BE TURNED ON A LATHE. JUST GIVE THE DIAMETER AND THE LENGTH.
MATERIALS LIST
TITLE BLOCK
GIVE THE MATERIAL SIZE, SHAPE, AND TYPE IN THE "MATERIALS LIST."

FIG. 5·41

Taper turning is similar to straight turning, except the finished stock will slope from a large diameter to a smaller diameter -- like a cut-off ice cream cone.

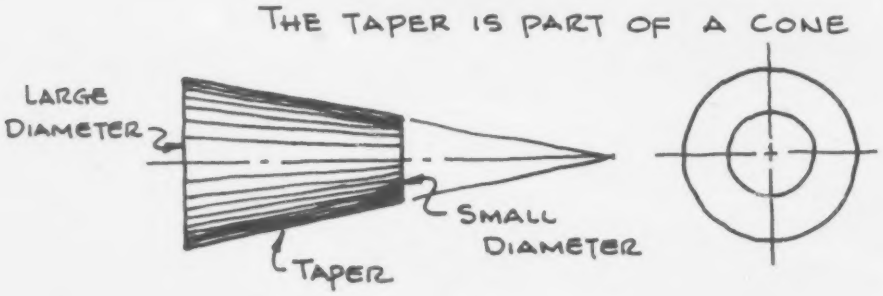


FIG. 5·42

Tapers are made by cutting more metal from one end of the stock than the other, obviously. The big problem is "How do you get a uniform taper?" There are four ways to handle this problem:

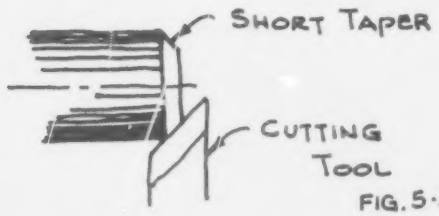


FIG. 5·43

1) Using a cutting tool with a preset cutting angle, called a form tool. This can only be used to cut short tapers, though, like a chamfer.

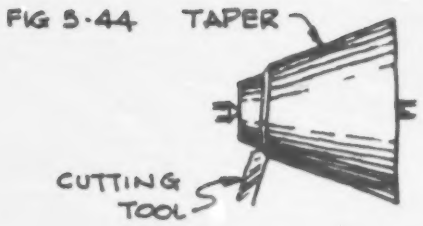


FIG 5-44

2) Offset the cutting tool to the desired taper angle and start cutting at the small-diameter end, cutting away an increasing amount of metal on each pass.

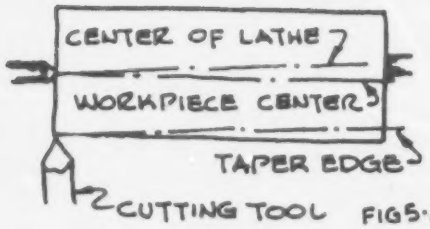


FIG5-45

3) Offset the stock so that it doesn't turn true -- it wobbles out of round. When the cutting tool is moved along the length of the stock, it will cut off more metal at the offset end, producing a taper.

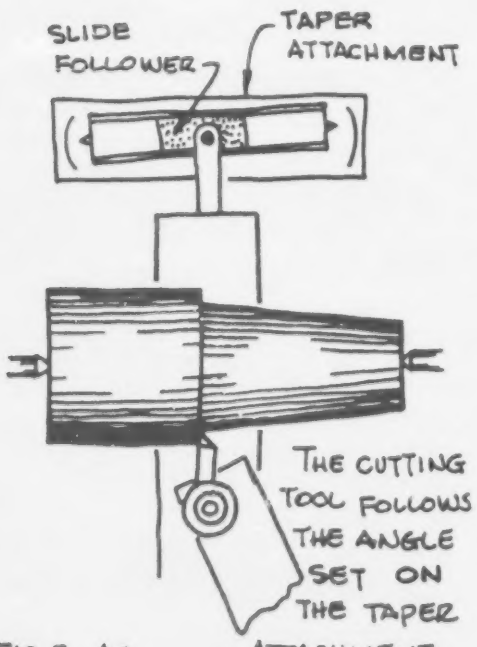


FIG. 5. 46

4) Use a special taper attachment on the lathe. This is the most accurate way to make a taper. The top of the attachment is marked in degrees on one side and inches per foot on the other. The cutting tool is attached to the top by a slide and follower mechanism, so the cutting tool is moved in or out as it passes along the length of the workpiece. It is this in or out motion that causes the taper to be formed.

When you dimension a taper, you have almost as many choices as when you make a taper. You have three choices:

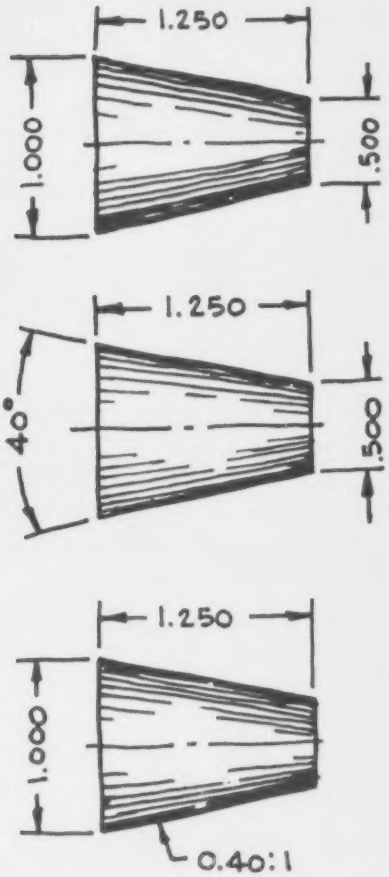


FIG 5·47

1) Give both diameters (large and small) and the length of the taper.

2) Give one diameter, the length of the taper, and the angle between the tapered sides of the workpiece (called the included angle).

3) Give one diameter, the length of the taper, and *note the taper as a ratio.* Before the start of the Metric conversion, tapers were noted as fractions of an inch per foot (1/4" per foot). Now they are given as a ratio of a decimal part of an inch to an inch, or a decimal part of a millimeter to a millimeter (.006:1).

Since a ratio is a comparison of one thing to another, it doesn't matter whether you are comparing millimeters or inches as long as both the decimal and the whole number are in the same unit of measurement.

Chamfers, short tapers made by a form tool, are usually found at the end of a shaft and/or at the mouth of a hole. They are used to help line up the shaft and the hole and prevent the edge of either one from becoming chipped, during the assembly operation.

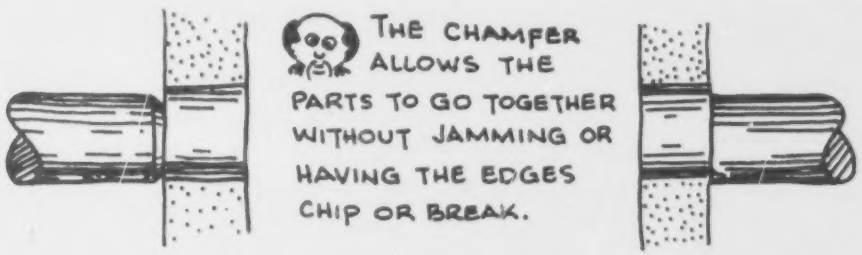


FIG. 5·48

Chamfers are dimensioned by giving the angle (usually 30°, 60°, or 45°) and the length in a local note; actually dimension the angle and length; or dimension the length and width of the chamfer.

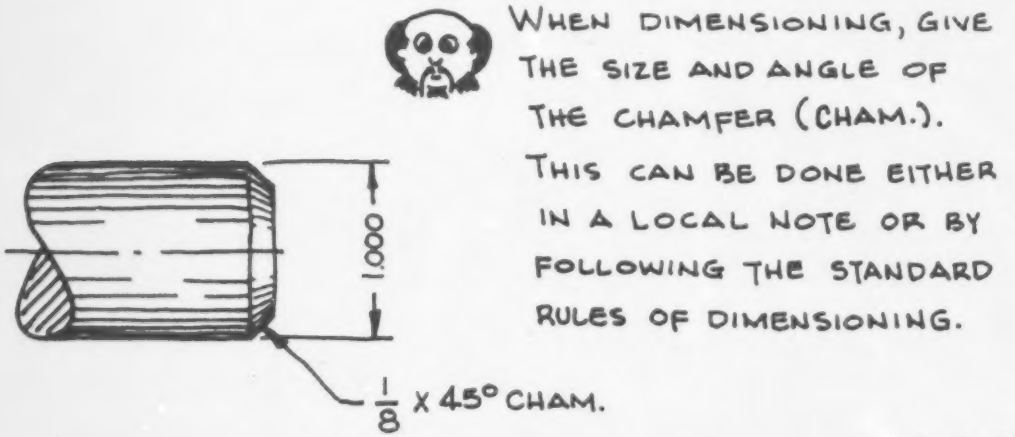

WHEN DIMENSIONING, GIVE THE SIZE AND ANGLE OF THE CHAMFER (CHAM.). THIS CAN BE DONE EITHER IN A LOCAL NOTE OR BY FOLLOWING THE STANDARD RULES OF DIMENSIONING.
1.00
1/8 X 45° CHAM.

FIG. 5-49

Knurling is a method of cutting a pattern into a workpiece —— either a straight or diamond pattern. Usually this is done on a knob to make it easier to grip; or on a shaft to give a large area of contact, but not actually press the two parts tightly together.

Straight knurling is a series of straight grooves around the edge of a workpiece (usually cylindrical), like the edge of a dime or quarter. Diamond knurling is a series of angled grooves, one set running in one direction and a second set running in the opposite direction, to form the diamond pattern. The grip of a ratchet has diamond knurling on it to provide a nonslip surface.

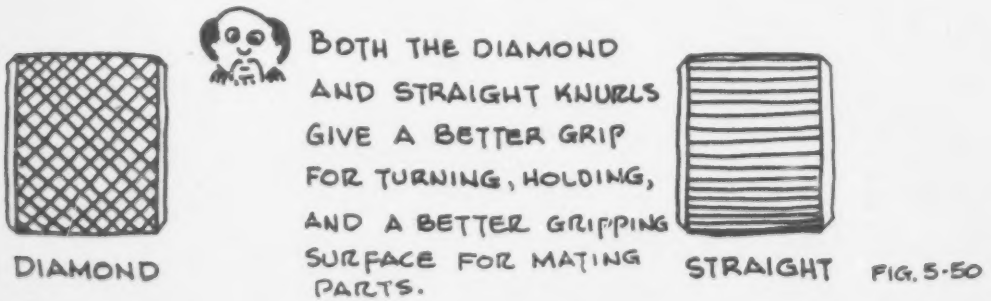

BOTH THE DIAMOND AND STRAIGHT KNURLS GIVE A BETTER GRIP FOR TURNING, HOLDING, AND A BETTER GRIPPING SURFACE FOR MATING PARTS.
DIAMOND
STRAIGHT

FIG. 5-50

The knurl pattern should be drawn over the entire area of a small part, but on a large part it is only necessary to draw enough to show where the operation begins and ends (the limits of the operation) and what pattern is to be used.

Dimensioning a knurl is done through a local note. The note would include either the pitch (the distance between the grooves) *or* the class of knurl (fine, medium, coarse) and the pattern type (straight or diamond).

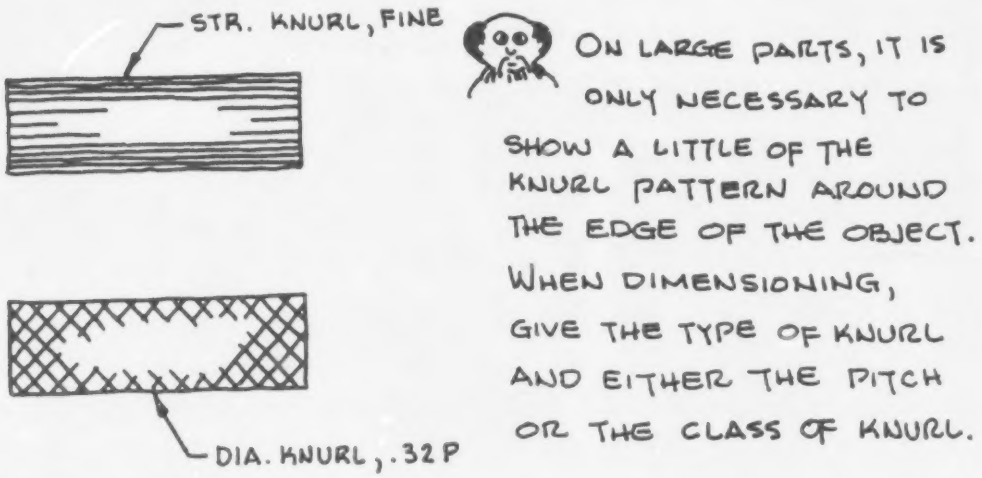

STR. KNURL, FINE
DIA. KNURL, .32 P
ON LARGE PARTS, IT IS ONLY NECESSARY TO SHOW A LITTLE OF THE KNURL PATTERN AROUND THE EDGE OF THE OBJECT.
WHEN DIMENSIONING, GIVE THE TYPE OF KNURL AND EITHER THE PITCH OR THE CLASS OF KNURL.

FIG 5-51

Threads are cut by moving the cutting tool forward at a specific speed to obtain the spiral form of the thread.

They are dimensioned by using a local note containing the diameter of the stock the threads are cut from (nominal size), the number of threads per inch, the thread series (standard used), the thread class (loose or tight fit), and whether the thread is internal (inside) or external (outside).

WHEN DIMENSIONING THREADS, USE A LOCAL NOTE AND GIVE THE NUMBER OF THREADS PER INCH (I.E. 16), THE TYPE OF THREAD OR THREAD FORM (I.E. UNC), THE CLASS OF FIT (I.E. 2), AND WHETHER THE THREADS ARE EXTERNAL (A) OR INTERNAL (B).

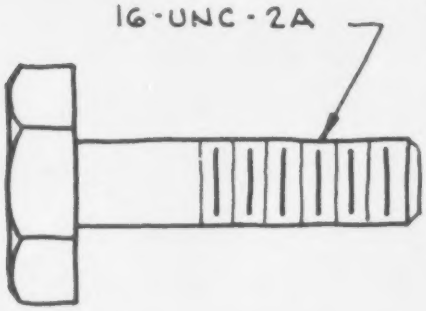


FIG. 5-52

MILLING

The milling machine uses a cutting/chipping action to remove large amounts of metal accurately. Rather than the workpiece spinning, like the lathe, the milling machine uses a spinning cutting blade (fixed) and the workpiece is fed (pushed) into the cutters. The cutters look like fat saw blades, over-sized drills, or variations of other common tools.

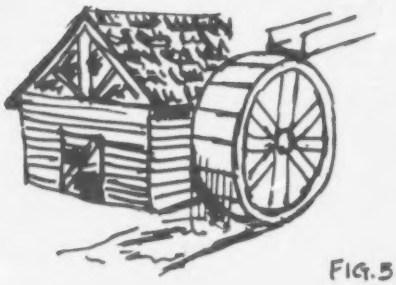
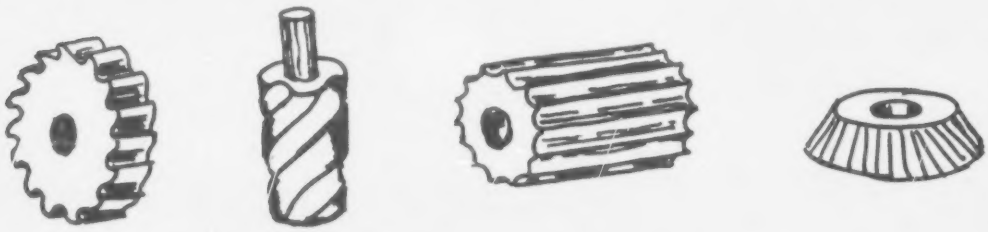


FIG. 5-53



MILLING CUTTERS COME IN A WIDE VARIETY OF SIZES AND SHAPES.

FIG 5-54

The milling machine is used to cut slots, form shapes (dovetails, round corners, etc.), make gears, and a wide variety of other operations, including thread cutting. The lathe and milling machine are the two most versatile and necessary pieces of equipment in the machine shop.

When dimensioning a shape cut by the milling machine, you don't specify how the operation is performed. The shape would be dimensioned using the rules of size, shape and location dimensioning given in Unit III.

REGARDLESS OF THE SIZE OR TYPE MILLING CUTTER, IT IS ONLY NECESSARY TO DIMENSION THE OBJECT. IT IS NOT NECESSARY TO SPECIFY THE MACHINE PROCESS OR CUTTER SIZE.

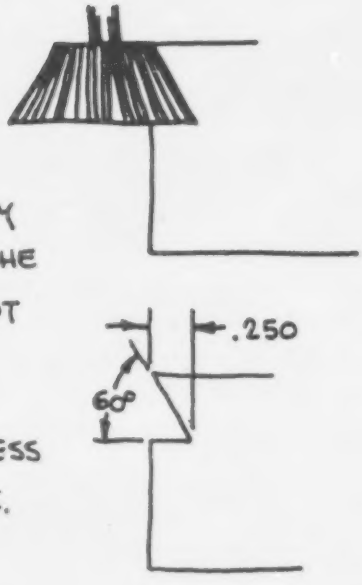


FIG. 5-55

PLANER AND SHAPER

The basic operation of the planer is to produce flat vertical, horizontal, or inclined surfaces. The shaper does almost the same operation, but can cut internal surfaces as well as external surfaces. Both have a single-point cutting tool (like the lathe) that chips metal away from the workpiece. The workpiece is fixed, and the cutting tool moves over the surface (like the milling machine), leveling the surface of the object. Generally, no dimensions or notes are needed, since this operation is a preparatory step in the total machine operation. However, in some cases it is necessary to say "how flat is flat". We will learn more about this in a later unit.

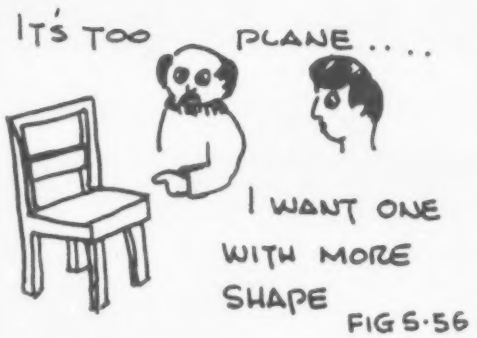


FIG 5·56

WHAT'S LEFT?

There are many machine operations and production processes we haven't talked about; we have only scratched the surface. You can form a work piece by sawing, filing, grinding, honing, lapping, drawing, polishing, and more. If you want to learn more about machine operations, visit a machine shop, talk to a machinist, or visit your library.

THINK ABOUT IT!

Look at the things around your house. Can you recognize how any of them were made?

The other processes for manufacturing items are numerous; see if you can list and explain five of them.

The effect of cost on the use of a specific operation is great. Can you explain why this is so and what the Federal Government does to ease this problem?

What effect do you think the metric conversion process will have on the supply of replacement parts for American-built machinery?

Considering drafting is a universal language and I.S.O. was developed to standardize drafting practices, how do you think machine processes compare throughout the world and why?

EXERCISE YOUR BRAIN

Why do you need to learn about machine processes?

Sketch a typical sand casting flask and label the parts.

Why are the patterns and cores used to make sand castings made larger than the finished product size?

What are fillets and rounds? Why are they used?

What is die casting and how does it differ from sand casting?

What is investment casting?

Explain the differences between cold rolling and hot rolling.

Explain forging and compare it to blacksmithing.

List and sketch the proper method of dimensioning seven (7) methods of producing holes.

The lathe and milling machines are two important pieces of machining equipment. Explain the economic basis of their importance.

Name four (4) machine operations which can be performed on the lathe. Sketch and dimension an example of each.

Explain the purpose and difference between the planer and shaper.

UNIT VI: SEQUENCE OF OPERATIONS

GET READY!

You have learned some standard dimensioning practices, a background in measurement, and some of the production processes. Now it's time to put all of it together and do something with it.

FIG 6-1

GET SET!

"Sequence of operations" means the order the machine operations take place. You have been told machine operations can determine how an object will be dimensioned; the example of dimensioning the hose nozzle that follows illustrates this point.

FIG 6-2

GO!

The first thing the machinist needs to know is how much material he is going to need to make the hose nozzle. The size of the material, called the stock size, is found in the parts list or bill of materials, along with the quantity needed and the type of material (steel, brass, etc.). Compare the given stock size with the overall size of the finished nozzle.

FIG 6-3

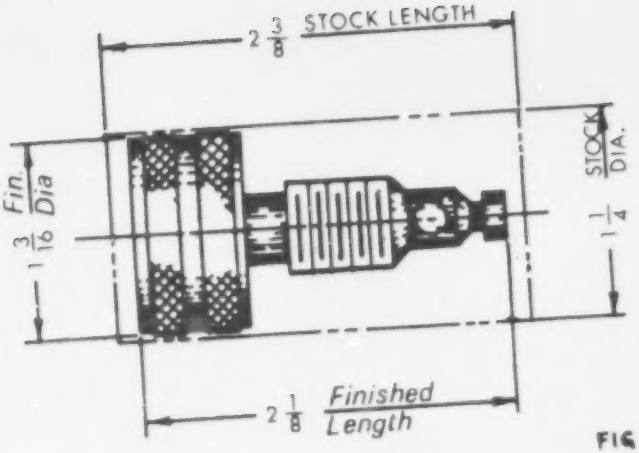


FIG 6-4

The nozzle will be made on the lathe, since it is a cylindrical object. The first machine operation that must be performed is to flatten one end of the stock and make it perpendicular to the sides. The second step is to machine 1½" of the stock to a .500" diameter. The facing operation (flattening of the end) is not dimensioned, but the turning down to the .500" diameter is dimensioned.

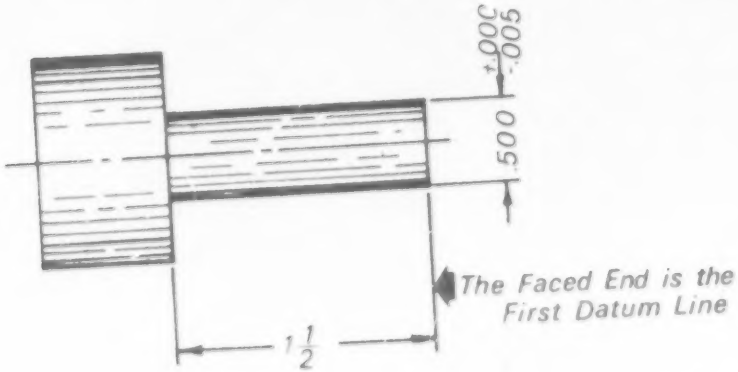


FIG 6-5

The third step is to turn the last .175" down to a .235" diameter. Take note of the fact that the faced end of the stock is used as the datum line for the dimensions given so far. The first machining operation determined where the datum line would be.

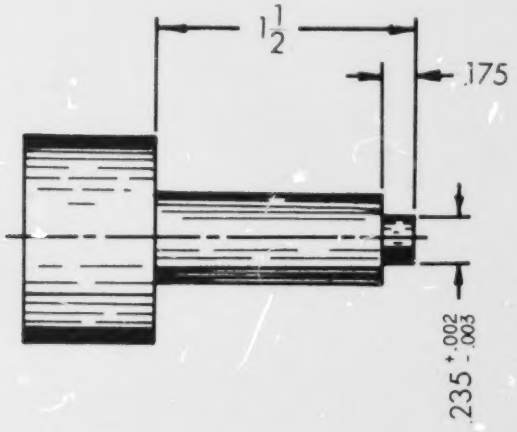


FIG 6·6

The fourth step is to cut a neck .065" wide. Take note that the datum line has changed. It is now .175" in from the end of the nozzle, the last operation to be performed.

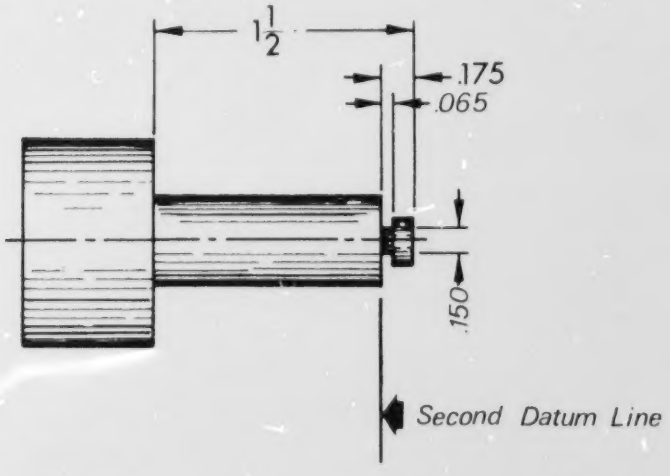


FIG 6·7

The fifth step is to measure in .700" from the end and turn the stock between the neck and the .700" measurement down to .340". Notice that since the machinist is measuring from the end, your datum line has shifted back to the faced end of the stock.

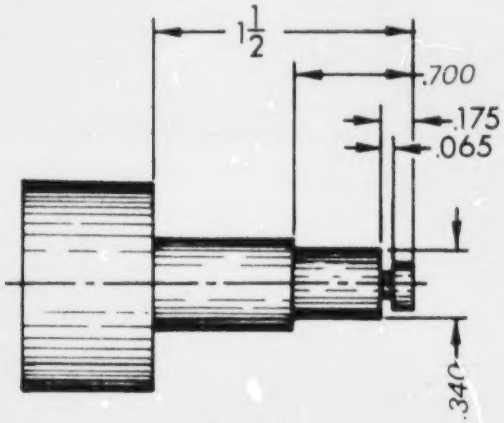


FIG 6-8

The sixth step is to cut two (2) chamfers, one between the neck and the .340" diameter; and the other between the .340" diameter and the .500" diameter. These are dimensioned by local notes.

The seventh step is to cut a 1/4" neck at the left end of the .500" diameter. Again, the datum line has changed from the faced end of the stock to the left end of the .500" diameter. The diameter of the neck is .400".

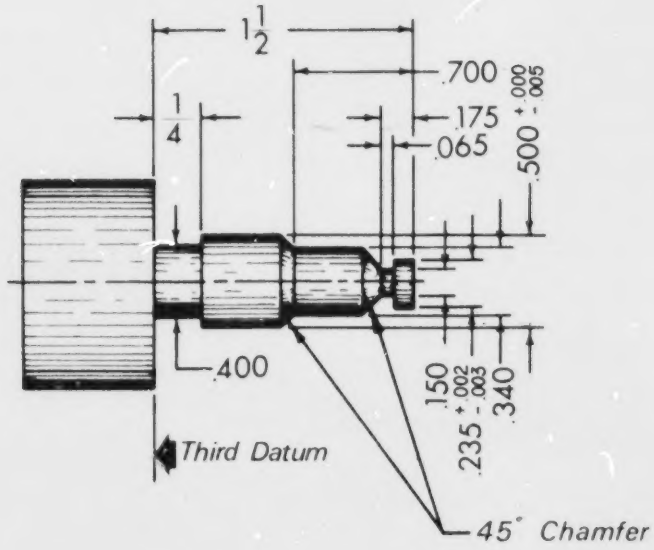


FIG 6·9

The eighth and ninth steps are to thread the .500" diameter part and drill a 3/16" diameter hole in the center of the .340" diameter part. The threads are "called out" in a note only. The 3/16" drilled hole is called out in a note, but must be located with a dimension using the left end of the .340" diameter as a datum line.

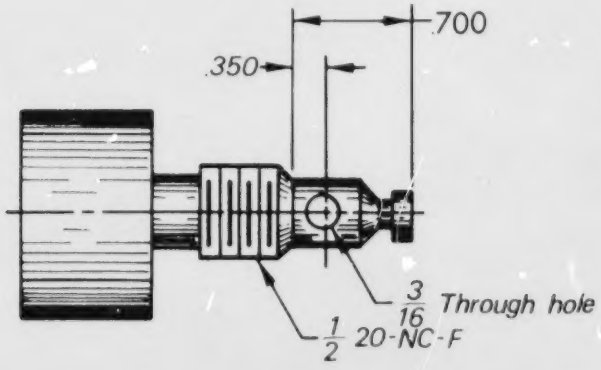


FIG 6·10

The tenth step is to face the 1–3/16" end to 5/8" length and cut a diamond knurl into the surface. The facing operation is not called out, but the finished length is given. The knurl is marked by a note and the diamond pattern drawn on the end of the object.

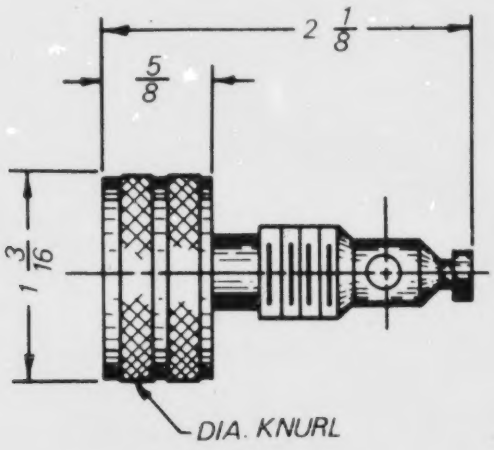


FIG 6·11

The eleventh step is to drill a 1/4" hole through to the 3/16" hole. This is drilled from the knurled end, and is called out in a note. This hole is used as a pilot for a 5/16" drilled hole 1–3/8" deep, which will be called out in the same note.

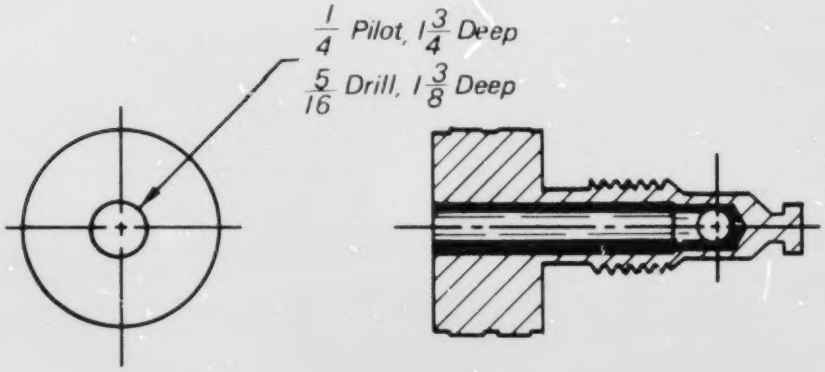


FIG 6·12

The end is then bored to a .975" inside diameter, .500" deep. A relief or undercut 1.075" in diameter and 1/8" wide is also made. Both operations are called out in one note, since it would be hard to dimension the undercut separately on a drawing this size. On large drawings, the undercut could be dimensioned separately.

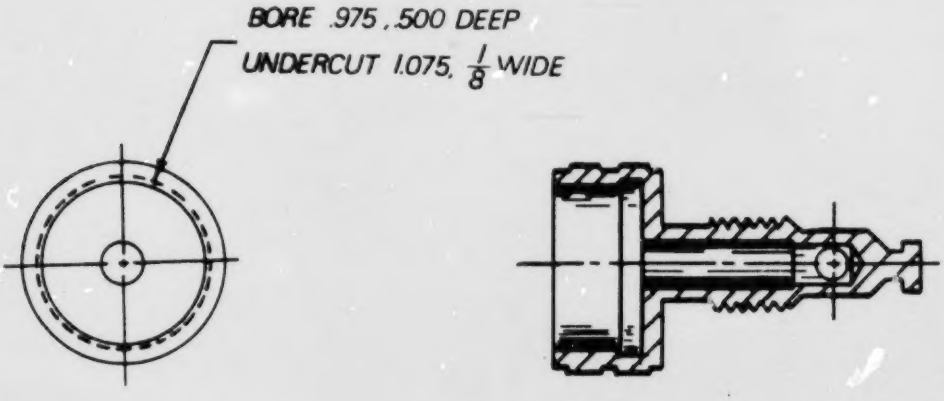


FIG 6·13

The final machine operation is to thread the bored part. Again, this is called out by a note. The only thing left to do is remove any burrs or sharp edges left by the machine operations. This will be covered by a general note.

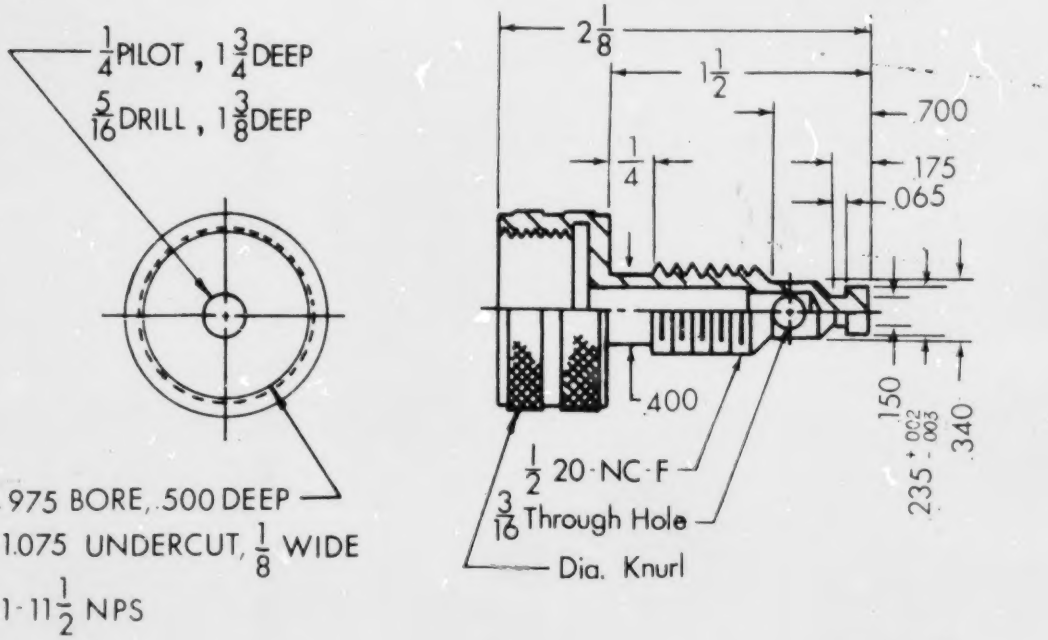


FIG 6-14

THINK ABOUT IT!

Look at several objects around the house or in your daily life and think about how and in what order they were produced.

Consider how dimensioning practices are affected by machine processes and sequence of operations. How are cost and scheduling affected by machine processes and sequence of operations?

Why is it necessary to develop a sequence of operations for the manufacturing of a product?

Write the sequence of operations for making a 1–1/2" American Standard Hexagon Head Bolt, 6" long. Make the bolt out of steel hexagon bar stock.

EXERCISE YOUR BRAIN!

Now, go back to page 89. Try to draw and dimension the adjusting sleeve for the nozzle dimensioned in this chapter.

1. Material size: 3/4 O.D. x 1–5/8 lg., aluminum.
2. Face one end and knurl 3/4" lg.
3. Drill 9/32 pilot hole.
4. Redrill with 7/16" drill, 1–5/16 deep. Drill from knurled end.
5. Thread knurled end (1/2 – 20 NF, 1 in. deep).
6. Face part to 1–9/16 length, from unfaced end.
7. Make 90° countersink so the 9/32 hole is 1/8 long.
8. Turn 10° taper, 9/16 long from knurled end.
9. Leave 1/8" straight on unknurled end and chamfer to small diameter of taper.

FINGERPRINTS

Each machine tool leaves marks on the surface of the workpiece, like fingerprints. These fingerprints are called machine finishes and are as distinctive as our own. The difference is that these fingerprints can be reproduced and ours can't.

FIG.7-1

It would be impossible to remove all these marks from the workpiece, and in many cases they are necessary. Also, like fingerprints, many of them are unnoticeable without a magnifying glass or microscope.

WHY WORRY?

Since we can't totally eliminate the tool marks, we live with them and use them to work for us. When there are deep cuts or grooves in the surface of a workpiece that is supposed to be polished, it can be costly. On the other hand, what happens when you count on a series of spiral grooves to distribute lubricating oil and they aren't there? As you can see, too good a finish could be worse than a finish that isn't good enough.

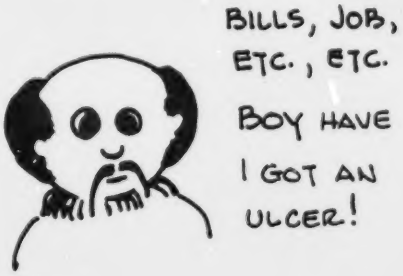


FIG.7-2

Another reason to worry about machine finishes is cost -- economics. The better or more accurate the finish, the more it costs. This is because an accurate finish requires more time, machining, and control. Some polished finishes have to be done by hand, requiring a great deal of time, people, and money.

CHECK!

FIG. 7-3

Checking the accuracy of a machine finish is just as costly as making the finish. The checking of a good finish requires even better test equipment. The more accurate the finish, the more complex the test equipment, and the more costly they both are. The checking of the accuracy of a workpiece (its size, finish, etc.) is called quality control.

Quality control of a rough surface can be done by eye, simply by comparing the finish of the workpiece with a sample finish. Even a good finish can be checked by comparing it to a sample under a microscope. More accurate finishes require optical testing -- measuring how much light is bent or bounced away by the marks, no matter how small, on the surface of a workpiece.

COMING TO TERMS

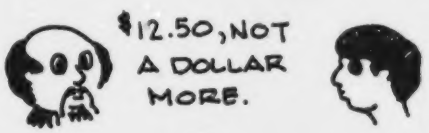
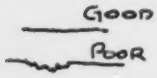


FIG. 7-4

Before going any further, it is necessary to understand and know some of the terms used when talking about machine finishes.



1. **Surface Quality** -- How smooth (a good or high surface quality) or rough (a poor or low surface quality) a machined part is.

2. **Controlled surface** -- A surface that is supposed to have a given surface quality. The engineer and design draftsman will decide whether the surface is to have a high, medium or low surface quality, and the machinist is required to produce that finish.

3. **Non-controlled surface** -- A surface that does not require an exact surface quality. Since all manufacturing processes produce a range of finishes unique to that process, the machinist must obtain a finish within that general range.

4. **Microinch** -- A microinch is one millionth of an inch (.000001 or 1/1,000,000). Roughness values are given in microinches, and they are the most common way to specify surface quality.

5. **Roughness** -- Roughness is the small peaks and valleys left by the cutting tool on the surface of the workpiece. The vertical distance or height from one valley to the nearest peak is called the roughness height. This is measured in microinches and is the most common method of controlling surface quality. A roughness number is given to the surface. This number stands for the average distance from a perfectly finished surface that the actual surface is.

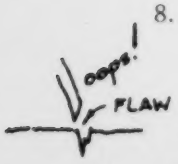
Roughness width is the distance from one peak to another. This distance is measured in tenths, thousandths, or ten-thousandths of an inch.

6. **Waviness** -- Waviness is the larger up-and-down changes caused by the shaking and vibrations of machinery. The waviness height is the average vertical distance of the large peaks. This is like measuring the perpendicular distance from the Mississippi Valley to the top of the Appalachian Mountains and to the top of the Rockies, then taking the average of the two. Waviness height is measured in tenths, hundredths, or thousandths of an inch, or even finer in some special cases.

Waviness width is the maximum distance allowed between large peaks, and is measured in whole and decimal parts of an inch. This is like measuring the horizontal distance between the Rockies and the Appalachian Mountains.

7. **Lay** -- This is the direction and pattern of the marks and grooves produced by the machine process. The lay is important to some functions and of no importance to others. Also, the needed pattern can sometimes determine which machine process should be used.

8. **Flaw** -- A flaw is a mark, groove, or break in the surface which is not wanted and is not a part of the normal machine-tool pattern. Flaws are caused by imperfections in the metal, a loose cutting tool, improper feeds, improper cutting speeds, etc.



These terms will be used in the next chapter to explain how to select and note a machine finish.

UNIT VIII: FINISH MARKS

MARK I

Three types of finish marks are used to point out a surface that is to be machined and tell what type finish is required. The first type is a sort of slanted "F" (See Fig. 8–1). This is the older way to call out a surface to be finished. Though this type is rapidly being replaced, you will run into it in older industrial drawings and textbooks. With a number beside it, the "F" represents a fine (F3), medium (F2), or coarse (F1) finish. A plain "F" can be used to represent a finish equal to the normal finish obtained from a particular machine process.

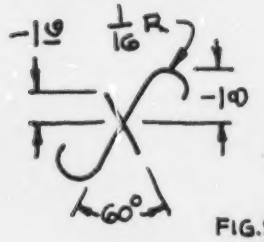


FIG. 8-1

MARK II

The second type of finish mark is the "V" (See Fig. 8–2). This is like an arrow pointing to the surface and prevents any confusion about which surface is to be machined. Again, the "V" without anything else added means the finish will be equal to whatever the normal machine finish will be.

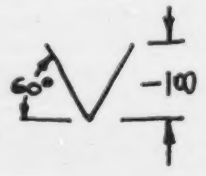


FIG 8-2

Letters can be placed within the opening of the "V" to specify a general range of finish (R for rough) or a particular machine finish (G for grind). Also, numbers can be placed in the "V" to call out a specific machine finish. The number would be equal to the desired roughness height.

The right side of the "V" is sometimes extended, so that it looks like a checkmark rather than a V.

MARK III

The third type of finish mark is a variation of the "V" finish mark. The right leg of the "V" is extended and a horizontal bar is added (See Fig. 8–3). This symbol is more appropriately called a surface-quality symbol.

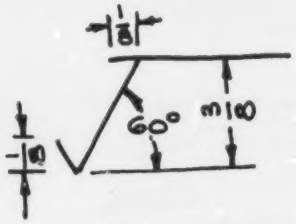


FIG. 8-3

The surface-quality symbol has the ability to give all the necessary information to control and check the finish and its quality.

The roughness height is added to the "V" part of the symbol as a numerical value and represents a number of microinches. For example, 63 stands for 63 microinches, .000063" or 63/1,000,000". The exact number used is determined by the function of the workpiece (what it's supposed to do) and the machining method used to make the workpiece.

The standard roughness heights used to specify a machine finish are:

1	6	20	63	200	600
2	8	25	80	250	800
3	10	32	100	320	1000
4	13	40	125	400	
5	16	50	160	500	

The most commonly used roughness heights are the 16, 32, 63, 125, 250, 500, and 1000.

The roughness height range for turning (lathe work) is from 16 to 125 microinches. The 16 and 32 are the most commonly used fine finishes, the 63 is a medium finish, and the 125 is the most commonly used coarser finish that is still in the medium range. The 250, 500, and 1000 are the most commonly used of the coarse finishes.

Suppose you have a cylindrical workpiece where the surface is to be used to spread a lubricant and yet not bind with another part attached to it. As you can see, the finish has to be good, but not too good. As we have stated, the 63 roughness height is at the low end of the medium range and will provide a good surface for both requirements. Another way to be sure the finish is correct is to take the high end of the fine range (32 microinches) and the low end of the medium range (63 microinches), and allow the machinist to fall between the two.

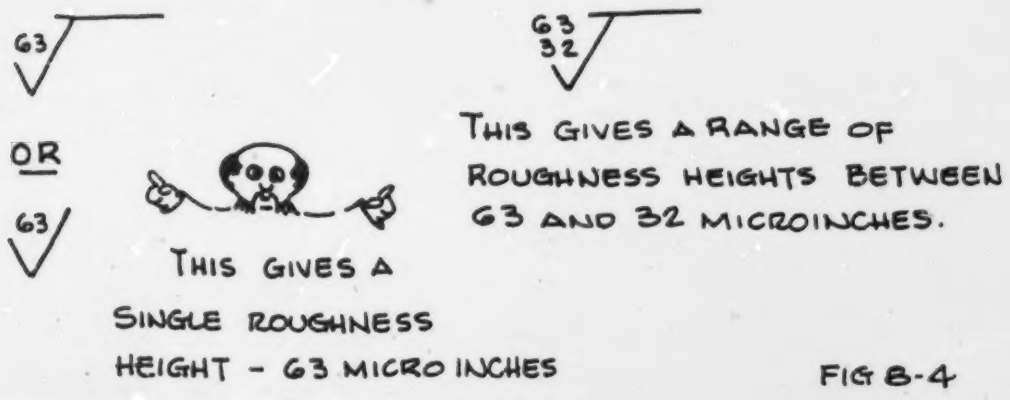


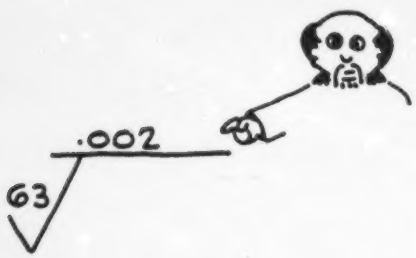
FIG 8-4

The waviness height is given above the horizontal bar and to the left. The standard values (in inches) for the waviness height are:

0.00002	0.0003	0.005
0.00003	0.0005	0.008
0.00005	0.0008	0.010
0.00008	0.001	0.015
0.0001	0.002	0.020
0.0002	0.003	0.030

There are no charts or tables to allow you to choose waviness height values based

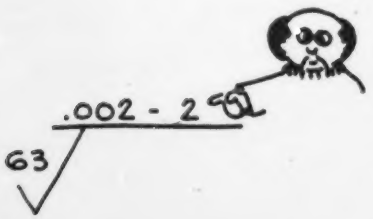
on function or manufacturing process, so you will have to judge which value to use. The ranges are 0.00002 to 0.00008 which is a very fine finish. 0.0001 to 0.0008 is a fine finish, 0.001 to 0.008 is a medium finish, and 0.010 to 0.030 is a coarse finish. The main thing to remember is that a fine roughness height value should have a fine waviness height value.



NOTE THAT 63 IS AT THE LOWER END OF THE MEDIUM ROUGHNESS-HEIGHT RANGE AND .002 IS AT THE LOWER END OF THE MEDIUM WAVINESS-HEIGHT RANGE.

FIG. 8-5

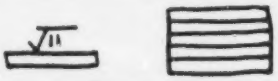
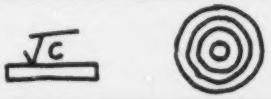
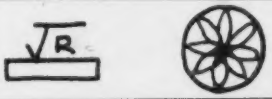
The maximum waviness width is given to the right of the waviness height value. This is measured in inches, and there are no standard tables or values for the waviness width. These values are usually set by the production engineer, and the finish is better when the values are larger.



DON'T GIVE A VALUE FOR THE WAVINESS WIDTH UNLESS THE ENGINEER TELLS YOU WHAT VALUE IS TO BE USED.

FIG. 8-6

The lay symbol is placed under the horizontal bar, near the point of the "V". There are six lay symbols that are used, and each is produced by a particular machine usage. These are also specified by the engineer, but they become easily recognizable after you become more experienced.

SYMBOL	MEANING	EXAMPLE
=	THE LAY WILL BE PARALLEL TO THE EDGE THE FINISH MARK IS DRAWN TO.	
⊥	THE LAY WILL BE PERPENDICULAR TO THE EDGE THE FINISH MARK IS DRAWN TO.	
C	THE LAY WILL FORM CONCENTRIC CIRCLES, WITH THE CENTER OF THE OBJECT AS THEIR CENTER.	
X	THE LAY WILL BE ANGULAR IN BOTH DIRECTIONS TO THE EDGE THE FINISH MARK IS DRAWN TO.	
M	THE LAY IS MULTIDIRECTIONAL, HAVING NO REAL PATTERN TO IT.	
R	THE LAY IS RADIAL, WITH ITS CENTER THE SAME AS THE OBJECT'S CENTER.	

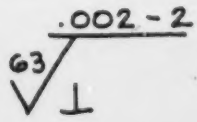


FIG. 8.7

The roughness-width cut-off is given directly below the horizontal bar, above the lay symbol. This value is decided upon by the engineer, and is given in inches. It is the particular length chosen to find the roughness height average or number. The most frequently used values for this length are 0.003, 0.010, 0.030, 0.100, 0.300, and 1.000.

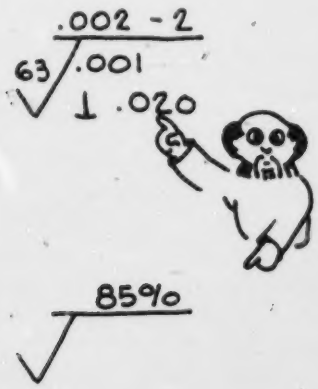
If you don't specify a value for the roughness-width cut-off, a value of 0.030 is automatically used to figure out the roughness height average.

WHEN THE ROUGHNESS-WIDTH CUT-OFF ISN'T GIVEN, A .030 VALUE WILL BE USED. IF YOU ARE IN DOUBT, DON'T GIVE A VALUE.

FIG. 8-8

The roughness width is given to the right of the lay symbol and below the roughness-width cut-off. Again this is measured in inches and will be specified by the engineer, since there aren't any standard tables or values.

The only other thing to be given with the surface-quality symbol is the percentage of minimum contact or bearing area. This is area needed for a proper fit between mating parts, or for the contact area between a shaft and ball bearing, etc. This is given by a percentage over the horizontal bar, in place of the waviness height and width.



THE ROUGHNESS WIDTH SHOULD ONLY BE GIVEN IF THE ENGINEER GIVES YOU THAT INFORMATION.

THE SAME THING IS TRUE FOR THE SURFACE QUALITY (PERCENTAGE).

FIG. 8-9

HOW MUCH?!

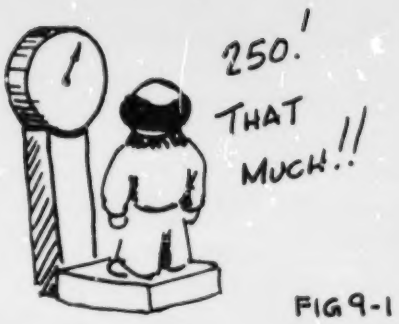


FIG 9-1

How many measurements are really exact? Only those that are taken from an object directly, and those are only as exact as the tool used to take the measurements. Even the tools used to make the object affect its size. For example, a carpenter is lucky to build a house where everything is accurate to within 1/16 of an inch (1/16 over or 1/16 under). The carpenter's rule is only divided into 16ths, and the width of a cutting blade is equal to a 16th. A small error here, another one there, and you soon have an error of a few inches. With a good carpenter, the errors will equal out -- some too long and some too short -- so that the final size will be accurate enough to prevent any major disaster.

You can see how errors can accumulate and be caused by cutting and measuring tools. The same thing happens in the machine shop. Even though the machinist can measure down to .0001 when necessary, it also means he can make greater errors. For the carpenter to be off by a 16th is not that bad, but for the machinist it is a disaster. A few thousandths can make the part worthless. What can we do to prevent the loss of valuable parts?

Tolerancing was developed for just this purpose. Tolerancing is a method of saying how large or small a part may be and still be useful. Most parts can be slightly larger or smaller than the stated size and still be able to work. Depending on the function of the part and how it is to be made, the tolerance will be high or low.

Before going further with the idea and methods of choosing tolerances, it is necessary to define some of the terms used in discussing tolerancing:

1. **Tolerance** -- As we mentioned before, a tolerance is how much a part is allowed to be off. Officially, a tolerance is the "total amount by which a given dimension may vary, or the difference between limits".[1]

2. **Limits** -- The limits are the largest and the smallest a workpiece can be and still work. Remember, both values are the limits.

[1] ANSI Y14.5 – 1966

3. **Nominal size** -- The nominal size is a value used to *identify* the size of an object. It may be in fractions or decimals. For example, a cold-finished, low-carbon steel round rod with a nominal size of 1 inch actually measures .998 of an inch, but it is easier to refer to it as a 1-inch rod -- the *nominal* size. In machine work, however, the nominal size is generally the same as the basic size, or the fractional equivalent of the basic size.

4. **Basic size** -- The basic size is the "Size from which the limits are derived by the application of the tolerances or allowances".[2] In other words, it is the starting point for the entire tolerancing process.

5. **Allowance** -- Allowance is "the minimum clearance or maximum interference intended between the maximum material limits of mating parts".[3] This would be the tightest possible fit between two parts, and still have the parts within the given limits. This can be found by taking the difference between the upper limit of the shaft and the lower limit of the hole.

Other terms will be defined in the following chapter, where you will learn to apply the concept of tolerancing. Before getting into the application of tolerances, you must decide how you are going to apply those tolerances.

HOW?!

There are many ways to apply tolerances to nominal or basic sizes. One is called fits -- the numbering of a tolerance and calling out, by number, of the tolerance. The part is dimensioned using the nominal size only. It is left up to the machinist and figure out what the minimum and maximum sizes are, based on the fit number.

FIG 9-2

NOMINAL SIZE AND THE TYPE FIT

[2] IBID
[3] IBID

The second method of applying tolerances is to dimension the object using the limits -- the largest and smallest allowable sizes of a workpiece. This method gives the machinist the dimensions he must work with, but doesn't give him the nominal or basic size.

The third method is the plus and minus method, which gives the basic size and the decimal values (positive or negative) needed to give the limits. Again, the machinist is required to do the majority of the work. This can create problems, since the decimal values can be added in one direction (+ *or* – value) only, or both directions (+ *and* – value) as the situation requires. If it is added in one direction only, it is called a unilateral tolerance. If it is added in both directions, it is called a bilateral tolerance. We will go into this in more detail later.

The fourth method is called general tolerances. General tolerances are tolerance values that hold true throughout the whole production process or throughout a majority of the processes. These tolerances are given in a general note and will be followed unless another tolerance is given on the drawing itself.

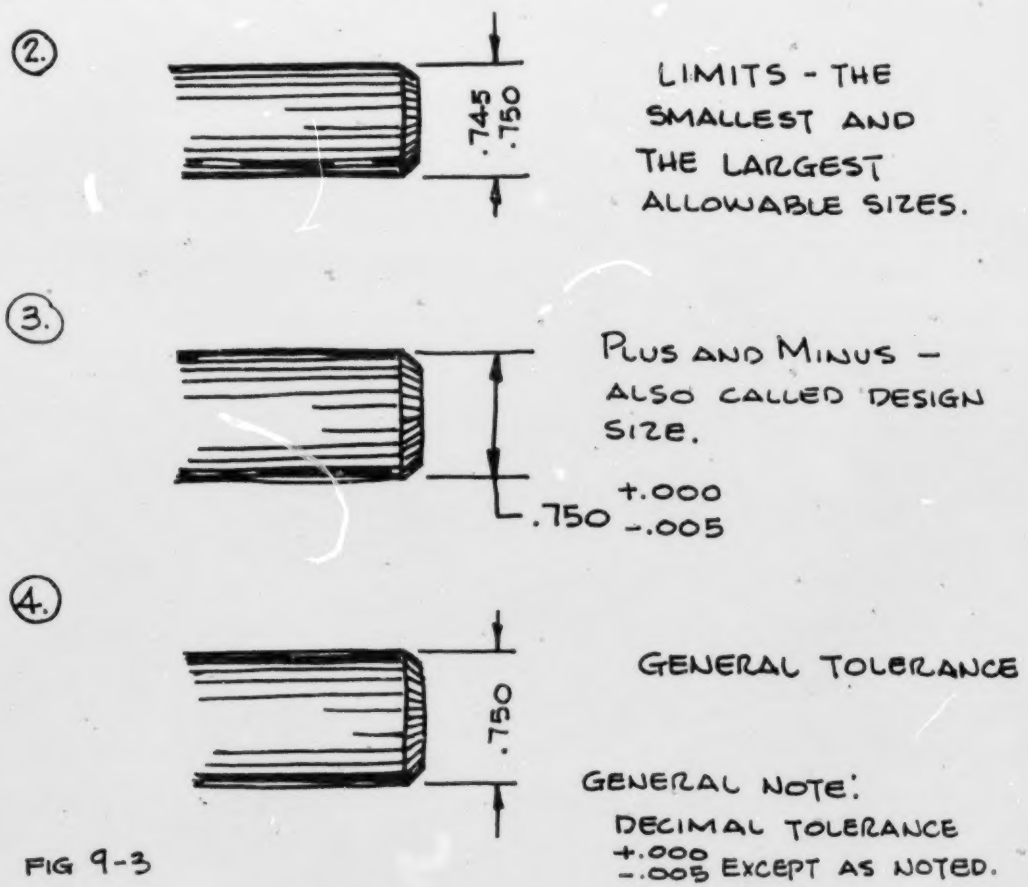


FIG 9-3

UNIT X: TOLERANCE APPLICATIONS

GENERAL

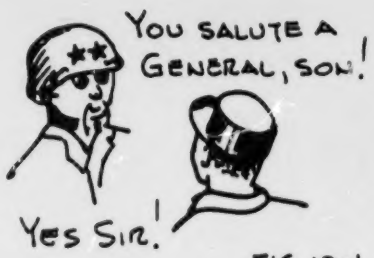


FIG 10-1

The application of a tolerance begins with the nominal size of the object. This value makes it easier to talk about the object, but means the machinist has a greater chance to make an error. Most nominal sizes are whole or fractions of an inch, and when the machinist is using a rule, rather than precision measuring instruments (micrometers, etc.), he has more of a chance to make a mistake. We are allowing for error in the measuring instrument as well as human error.

First, change the nominal size into a basic size -- convert it from a fraction to a decimal. The decimal shouldn't contain more or fewer digits to the right of the decimal point than the tolerance.

The tolerance is added to the basic size, and you have what is called the design size. For example, a shaft has a 1½" nominal diameter. Its basic size is 1.500, and the engineer set a tolerance of + or - .001. The design size would then be 1.500 ± .001.

The limits (the largest and smallest allowable sizes) would be obtained by adding .001 to 1.500 and subtracting .001 from 1.500. The results are 1.501 and 1.499. The 1.501 value is called the upper limit and the 1.499 value is called the lower limit.

To find the allowance you would need to know the limits for *two mating parts*. In the above example we have a shaft with an upper limit of 1.501 and a lower limit of 1.499. Suppose we wanted that shaft to fit into a hole with an upper limit of 1.505 and a lower limit of 1.504; then the allowance would be the difference between the largest possible shaft and the smallest possible hole. In this case, the largest possible shaft is 1.501 (the upper shaft limit) and the smallest possible hole is 1.504 (the lower hole limit). The difference is .003 and is called the allowance.

STANDARDS

We mentioned briefly in the last chapter that some machine processes have standard tolerances established for them. As a reference to be used on your drawings, here are some of the standard tolerances:

FIG 10-2

1. Dimensions which don't affect the function of the part:

   (a) 0 to 6 in.: ± 1/64

   (b) 6 to 18 in: ± 1/32

   (c) 18 in. plus: ± 1/16

2. Sand Castings:

   (a) Small: ± 1/32

   (b) Medium: ± 1/16

   (c) Large: increase to suit size

3. Die Casting:

   (a) Small and medium: ± 1/64 or less

   (b) Large: increase slightly

4. Forgings:

   (a) 1 lb. or less: ± 1/32

   (b) 1 lb. to 10 lbs.: ± 1/16

   (c) Up to 60 lbs.: ± 1/8

5. Drilling:

   (a) Size 60 to 30: +0.002, -0.000

   (b) Size 29 to 1: +0.004, -0.000

   (c) 1/4 to 1/2: +0.005, -0.000

   (d) 1/2 to 3/4: +0.008, -0.000

   (e) 3/4 to 1: +0.010, -0.000

   (f) 1 to 2: + 0.015, -0.000

6. Reaming:

   (a) Up to 1/2 dia.: +0.0005, -.0000

   (b) 1/2 to 1 dia.: +0.001, -0.000

   (c) 1 dia. and up: +0.0015, -0.0000

7. Lathe (Rough Turning):

   (a) 1/4 to 1/2 dia.: Total tol. 0.005

   (b) 1/2 to 1 dia.: Total tol. 0.007

   (c) 1 to 2 dia.: Total tol. 0.010

   (d) 2 dia. and up: Total tol. 0.015

8. Lathe (Finish Turning):

   (a) 1/4 to 1/2 dia.: Total tol. 0.002

   (b) 1/2 to 1 dia.: Total tol. 0.003

   (c) 1 dia. to 2 dia.: Total tol. 0.005

   (d) 2 dia. and up: Total tol. 0.007

9. Milling:

   (a) One surface at a time: Total tol. 0.002 to 0.003

   (b) Two surfaces or more at a time: Most important surface, total tol. 0.002; remaining surfaces, total tol. on each, 0.005.

10. Planing and Shaping:

    Used primarily on large parts only, total tol. of 0.005 to 0.010.

11. Broaching:

    (a) Up to 1 dia.: Total tol. 0.001

    (b) 1 to 2 dia.: Total tol. 0.002

    (c) 2 to 4 dia.: Total tol. 0.003

    (d) Surfaces up to 1 in apart: Total tol., 0.002

    (e) Surfaces 1 to 4 in apart: Total tol., 0.003

    (f) Surfaces 4 and over apart: Total tol., 0.004

12. Grinding:

    (a) Cylindrical: Total tol., 0.0005

    (b) Surface (flat): Total tol., 0.0005

13. Threads:

    (a) Thread tolerances vary with thread series, type, and class.

    Check *Machinery's Handbook* for exact tolerances.

The tolerances listed are the best possible tolerances using normal shop practices. The term "total tolerance" means the total of the values added to the basic size or subtracted from it. For example, a ground surface could have a tolerance of +0.0005 and −0.0000, or +0.00025 and −0.00025, or +0.0000 and −0.0005, or any other combination as long as the two values *total* 0.0005.

PLUS – MINUS

Tolerances can be given in a number of ways, as stated in the last chapter. The most widely used method is the plus–minus method. Actually there are two types of plus–minus tolerances. The first type is called unilateral tolerancing. This means "one direction", and the total tolerance value is either subtracted *or* added to the basic size, but not both.

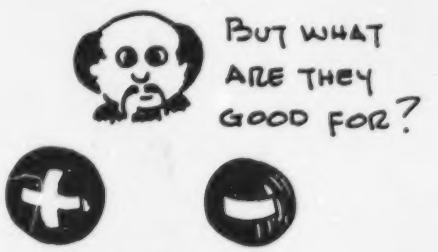


FIG 10-3

Unilateral tolerancing example: 1.500 basic size, .005 total tolerance.

$$1.500 \begin{matrix} +\ .005 \\ -\ .000 \end{matrix} \quad \text{or} \quad 1.500 \begin{matrix} +\ .000 \\ -\ .005 \end{matrix}$$

The second type is called bilateral tolerancing. This means "two directions"; *half* the total tolerance value is *both* added *and* subtracted from the basic size.

Bilateral tolerancing example: 1.500 basic size, .005 total tolerance.

$$1.500 \begin{matrix} +\ .0025 \text{ — 1/2 total tolerance} \\ -\ .0025 \text{ — 1/2 total tolerance} \end{matrix}$$

LIMITS

Limit dimensioning uses the largest and smallest possible sizes of an object as a form of tolerancing. The diameter of a hole or a shaft is dimensioned by giving the smallest possible size (lower limit) over the largest possible size (upper limit). The limits are found by adding or subtracting the tolerance from the basic size.

① $1\frac{1}{2}$ (NOMINAL SIZE) = 1.500 (BASIC SIZE)

② 1.500 (BASIC SIZE) $\pm$ .005 (TOLERANCE) = DESIGN SIZE

③ 1.500 (BASIC SIZE) + .005 = 1.505 (UPPER LIMIT)

1.500 (BASIC SIZE) − .005 = 1.495 (LOWER LIMIT)

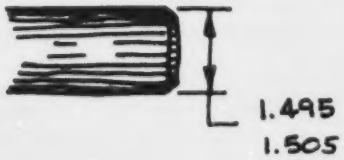


FIG 10-4

FITS

"Fit" refers to mating parts and how they go together. There are three types of fits: clearance, transition and interference. A *clearance fit* is where two parts go together without force. The two parts may only miss touching by one ten-thousandth of an inch or less, but as long as they still clear each other, it's a clearance fit. To check if a fit is a clearance fit, subtract the largest possible shaft size (upper shaft limit) from the smallest possible hole size (lower hole limit). If the difference (allowance) is a positive number (greater than zero), the fit is a clearance fit.

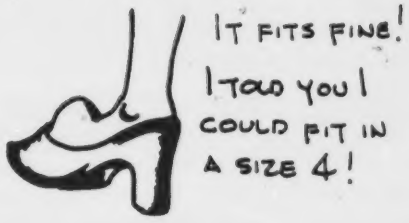


FIG10-5

Shaft:	1.500	+.000 -.005	Upper Limit:	1.500	1.505 - 1.500	
Hole:	1.505	+.005 -.000	Lower Limit:	1.505	0.005	allowance

(Greater than zero = clearance fit.)

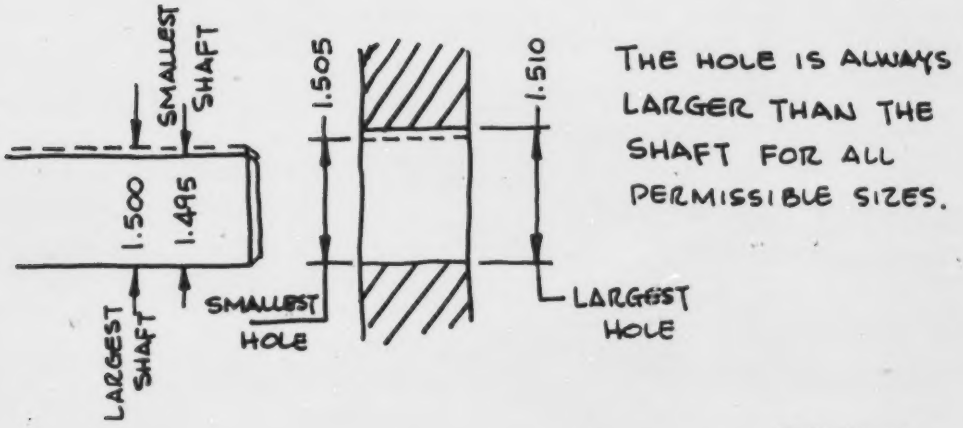


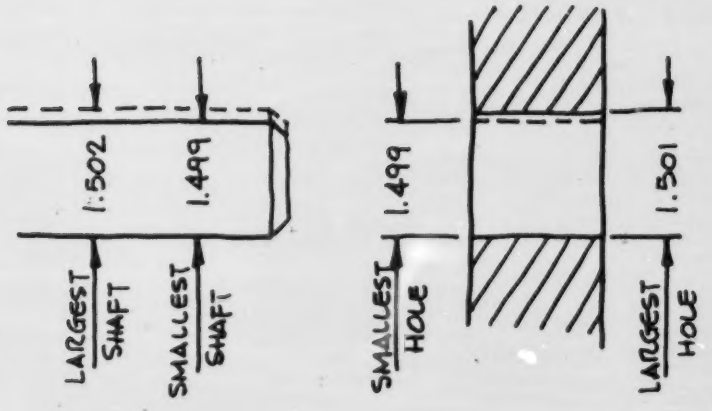
FIG. 10-6

A *transition fit* is when the two parts may or may not go together freely. If the shaft is made to the smallest allowable size and the hole to the largest possible size, the difference between the hole and the shaft will be a positive number. *This is not the allowance*; it is the clearance. When you figure out the allowance (smallest hole minus largest shaft), it will be a negative number (less than zero) or zero.

Shaft:	1.500	+.002 -.001	Lower limit: 1.499 Upper limit: 1.502
Hole:	1.500	±.001	Lower limit: 1.499 Upper limit: 1.501

Upper limit hole:	1.501	Lower limit hole:	1.499
− Lower limit shaft:	1.499	− Upper limit shaft:	1.502
Clearance:	0.002	Allowance:	−0.003
(Greater than zero)		(Less than zero)	

= Transition Fit



THE HOLE MAY OR MAY NOT BE LARGER THAN THE SHAFT, DEPENDING ON HOW THEY ARE MADE.

FIG. 10-7

An *interference fit* is one that will require force, sometimes a great deal of force, to put the two parts together. Both the clearance and allowance will be negative. Interference fits get their name from the fact that the metal of the shaft "interferes" with the metal around the hole.

Shaft: 1.500 $^{+.005}_{-.000}$

Upper shaft limit: 1.505
Lower shaft limit: 1.500

Hole: 1.495 ±.002

Upper hole limit: 1.497
Lower hole limit: 1.493

Upper hole limit:	1.497
− Lower shaft limit:	1.500
Clearance:	−0.003
(Less Than Zero)	

Lower hole limit:	1.493
− Upper shaft limit:	1.505
Allowance:	−0.012
(Less Than Zero)	

= Interference Fit

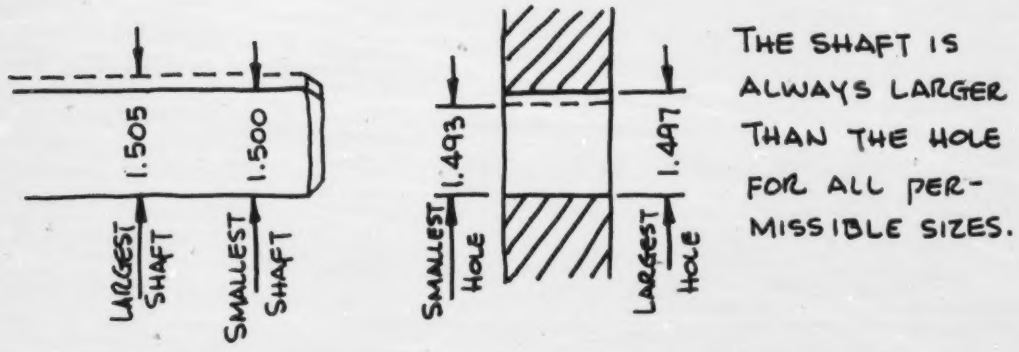


FIG. 10-8

These three types of fits are broken up into categories, and the clearances, allowances, and tolerances are standardized. This is done so tolerances can be reproduced whenever necessary. There are five classes of standard fits: (1) running and sliding fits (RC), (2) locational clearance fits (LC), (3) locational transition fits (LT), (4) locational interference fits (LN), and (5) force or shrink fits (FN).

RUNNING AND SLIDING

Running and sliding fits are all clearance fits. There are nine groups of running and sliding fits and they are designed to allow for lubrication, varying degrees of movement, and locating the parts.

FIG 10-9

RC 1 –This class has the least amount of clearance and the two parts have almost no play in them. They are used for high-precision work. (Example: Gauges)

RC 2 –This fit allows more clearance but still produces accurate locations. The parts will move and turn easily, but are not meant to move constantly. If they are kept in motion, they will bind up from the friction heat.

RC 3 –This fit will move freely and can be kept in motion without its binding. This is the least amount of clearance needed for the part to be kept in motion (running) and not bind up. It is used for oil-lubricated bearings, run at slow speeds with light pressure put on them.

RC 4 –This gives an accurate location and minimum play, and is used for grease- or oil-lubricated bearings. They will be run at medium speed and with medium pressure put on them.

RC 5 and RC 6 – These are the middle classes and are used in moving parts that are cold when they are started up and build up heat during their running.

RC 7 –This fit does not give a great deal of accuracy and should be used in cases where there are large changes in temperature.

RC 8 and RC 9 – These are loose fits and are used where parts are made using no precision machining (Example: rolled threads).

LOCATIONAL CLEARANCE

There are eleven locational clearance fits; they are used when two parts fit together but aren't going to move. The parts will fit together snugly and can be put together or taken apart easily.

FIG 10·10

LC 1 to LC 4 – They have a zero to near-zero clearance, but will almost always have a positive clearance. LC 1 and LC 2 may also be used as sliding fits.

LC 5 and LC 6 – These have minimum clearances used to locate non-moving parts. LC 5 can be used in place of RC 2, and LC 6 can be used in place of the RC 5 and RC 6 fits. This is done to give a closer tolerance on the parts.

LC 7 to LC 11 – These have increasingly larger clearances and tolerances and are used for bolt and nut assemblies, etc.

LOCATIONAL TRANSITION

There are six locational transition fits; they are used where location is important, but where there can be a small amount of clearance or interference.

FIG 10·11

LT 1 and LT 2 – The average of the limits for these fits give a slight clearance, less than the LC 1 to LC 3 fits. A slight interference should be allowable, and assembly would require pressure or light hammering.

LT 3 and LT 4 – These fits have an average clearance equal to zero, though in theory there is a slight clearance. They are used to eliminate movement caused by vibration (Example: keys). The parts are assembled by moderate pressure or hammering.

LT 5 and LT 6 – These fits have a slight interference on the average. These fits will require either force to assemble or selective assembly. They are used to prevent movement during heavy vibration and usage.

LOCATIONAL INTERFERENCE

Locational interference fits are used to accurately assemble and locate mating parts. When put together, the parts are held tightly together, but can be taken apart again, if necessary. There are six classes of locational interference fits.

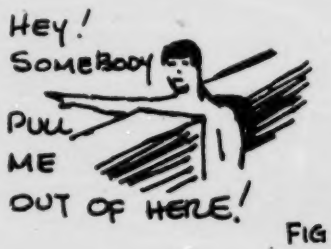


FIG 10-12

LN 1 and LN 2 – These fits have a minimum amount of interference and require the use of a small press to assemble. The parts can be assembled and disassembled with little problem. LN 1 and LN 2 fits are used to assemble dowel pins, etc.

LN 3 – This fit requires the use of a heavy press to assemble the parts. The amount of interference is greater than the LN 1 or LN 2 fits.

LN 4 to LN 6 – These fits can be used for permanent assembly of parts, but are meant to be used for soft metals (brass), light alloys (aluminum), or rigid plastics. The assembly will require the use of a heavy press.

FORCE OR SHRINK

Force or shrink fits are a special type of interference fit. The difference between minimum and maximum interference is slight, and this gives a constant pressure on the shaft. The parts are generally permanently assembled.

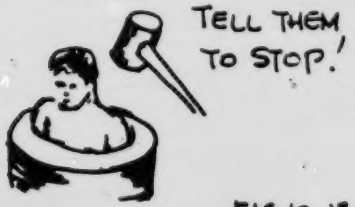


FIG 10-13

FN 1 – This is a *light drive fit* and requires light pressure to assemble. Once assembled, the parts can be disassembled, but one or both of the parts would probably have to be replaced.

FN 2 – This is a *medium drive fit* and can be used on average-size steel parts. This is also the tightest fit that can be used on cast iron parts.

FN 3 – This is a *heavy drive fit* and is used for heavy steel parts.

FN 4 and FN 5 – These are *force fits* and require a great deal of pressure to assemble the parts. Also, the FN 4 and FN 5 fits can be assembled by shrinking (cooling) the shaft. These are used on highly stressed materials like tool steel.

APPLICATIONS

Fits can be applied as tolerances in two ways, the basic hole system or the basic shaft system. Under normal conditions, the basic hole system will be used. In this system the hole will be dimensioned as the basic size, and the limits will be developed from there.

An example would be a hole and a shaft with a nominal size of 1½ inches. The hole would then have a basic size of 1.500. It is now necessary to check for the correct ANSI Standard Fit in the latest edition of *Machinery's Handbook*, or another standard reference. Part of the table for standard fits has been reproduced on the next page. Let's say the part is to have an RC 5 fit. When we look up RC 5, we see the size is given as a range rather than an exact size. We are looking for a range that will include 1.500, and this is 1.19 – 1.97. The "limits of clearance" are listed as 2.0 and 4.6. They're measured in thousandths of an inch, so the limits of clearance are .002 and .0046.

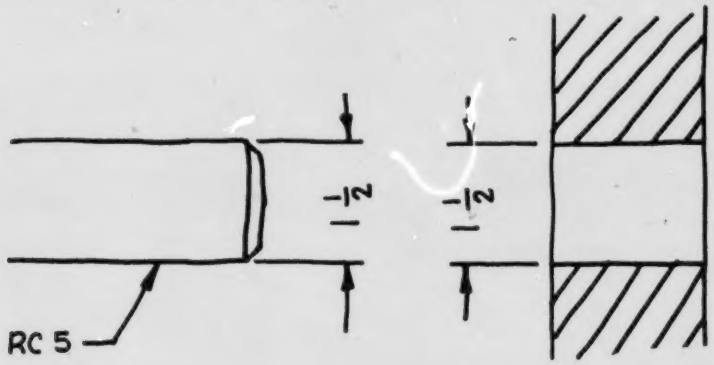


FIG. 10-14

Nominal Size Range Inches Over	To	Class RC 5 Limits of Clearance	Standard Limits Hole H7	Standard Limits Shaft e7
0	0.12	0.6 1.6	+0.6 −0	−0.6 −1.0
0.12	0.24	0.8 2.0	+0.7 −0	−0.8 −1.3
0.24	0.40	1.0 2.5	+0.9 −0	−1.0 −1.6
0.40	0.71	1.2 2.9	+1.0 −0	−1.2 −1.9
0.71	1.19	1.6 3.6	+1.2 −0	−1.6 −2.4
1.19	1.97	2.0 4.6	+1.6 −0	−2.0 −3.0
1.97	3.15	2.5 5.5	+1.8 −0	−2.5 −3.7
3.15	4.73	3.0 6.6	+2.2 −0	−3.0 −4.4
4.73	7.09	3.5 7.6	+2.5 −0	−3.5 −5.1

The limits of clearance are used to calculate the basic size of the shaft. Notice that the lower value for the limits of clearance is the same as the minus value for the shaft (2.0 and − 2.0). By subtracting the minimum clearance from the basic hole size, we get the upper limits of the shaft.

Basic Hole Size:	1.500
	− .002
Upper Shaft Limit	1.498

The remaining values of the chart are also given in thousandths of an inch and must be changed into decimals before they are used. The tolerance on the hole is −0 thousandths and +1.6 thousandths. These are equal to 0.000 and 0.0016. When added to the basic size of the hole, they will give the limits of the hole.

1.500	Basic Hole Size	1.5000	Basic Hole Size
−0.000		+0.0016	
1.500	Lower Hole Limit	1.5016	Upper Hole Limit

The standard limits of the shaft are −2.0 and −3.0, which are equal to −.002 and −.003. The −.002 has already been subtracted to get the upper shaft size, but if you are in doubt as to the minimum clearance size, use the smallest standard shaft limit. The other limit is subtracted from the *basic hole size* to get the lower shaft limit.

1.500	Basic Hole Size	1.500	Basic Hole Size
−0.002		−0.003	
1.498	Upper Shaft Limit	1.497	Lower Shaft Limit

The shaft and hole are then dimensioned using the limits or plus/minus dimensioning.

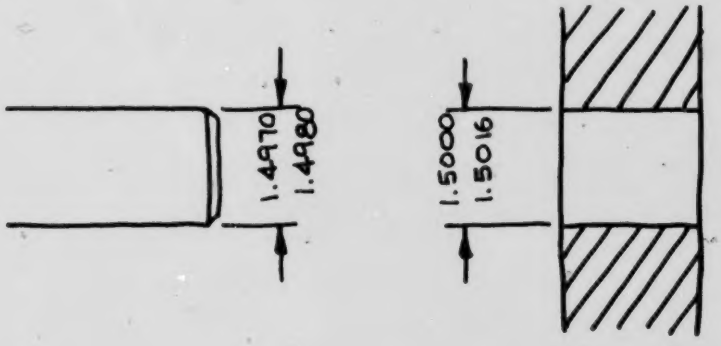


FIG. 10-15

Appendix

ABBREVIATIONS USED ON DRAWINGS

A

Absolute	ABS
Accelerate	ACCEL
Access Panel	AP
Accessory	ACCESS.
Account	ACCT
Accumulator	ACC
Actual	ACT.
Adapter	ADPT
Addendum	ADD.
Adjust	ADJ
Advance	ADV
Aeronautic	AERO
Aeronautical Material Specifications	AMS
Aeronautical Standards Group	ASG
After	AFT.
Aggregate	AGGR
Air Condition	AIR COND
Aircraft	ACFT
Air Force—Navy	AN
Airforce—Navy Aeronautical	ANA
Allowance	ALLOW.
Alloy	ALY
Alternate	ALT
Alternating Current	AC
Altitude	ALT
Aluminum	AL
Ambient	AMB
American Standard	AMER STD
American Standard Elevator Code	ASEC
American Wire Gage	AWG
Ammeter	AM.
Amount	AMT
Ampere	AMP
Amphibian—Amphibious	AMPH
Amplifier	AMPL
Amplitude Modulation	AM
Anneal	ANL
Antenna	ANT.
Anti-Aircraft	AA
Anti-Friction Bearing	AFB
Apparatus	APPAR
Apparatus	APP
Approved	APPD
Approximate	APPROX.
Arc Weld	ARC/W
Area	A
Armament	ARMT
Armature	ARM.
Armour Plate	ARM PL
Army Navy Airforce	ANAF
Arrangement	ARR.
Asbestos	ASB
Asphalt	ASPH
Assemble	ASSEM
Assembly	ASSY
Assistant	ASST
Associate	ASSOC
Association	ASSN
Atomic	AT
Atomic Hydrogen Weld	AT/W
Audible	AUD
Audio Frequency	AF
Authorized	AUTH
Automatic	AUTO.
Automatic Frequency Control	AFC
Automatic Voltage Control	AVC
Auxiliary	AUX
Avenue	AVE
Average	AVG
Aviation	AVN
Avoirdupois	AVDP

B

Babbitt	BAB
Back Feed	BF
Back Pressure	BP
Back to Back	B to B
Backgear	BG
Balance	BAL
Ball Bearing	BB
Barrel	BBL
Base Line	BL
Base Plate	BP
Basic	BSC
Battery	BAT. or BT
Baume'	BE.
Bearing	BRG
Beat Frequency Oscillator	BFO
Bell and Flange	B&F
Bell and Spigot	B&S
Bench Mark	BM
Bending Moment	M
Between	BET.
Between Centers	BC
Bevel	BEV
Bill of Material	B/M
Birmingham Wire Gage	BWG
Blank	BLK
Board	BD
Boiler	BLR
Bolster	BOLS
Bolt Circle	BC
Both Faces	BF
Both Sides	BS
Both Ways	BW
Bottom	BOT
Bottom Face	BF
Bracket	BRKT
Brake	BK
Brake Horsepower	BHP
Brass	BRS
Brazing	BRZG
Break	BRK
Brinell Hardness	BH
British Standard	BR STD
British Thermal Units	BTU
Broach	BRO
Bronze	BRZ
Brown & Sharpe	B&S
Building	BLDG
Building Line	BL
Bulkhead	BHD
Bureau. of Standards	BU STD
Bushing	BUSH.
Button	BUT.
By-Pass	BYP

C

Cabinet	CAB.
Cab Over Engine	COE
Cadmium Plate	CD PL
Calculate	CALC
Caliber	CAL
Candlepower	CP
Capacitance	C
Capacity	CAP.
Cap Screw	CAP SCR
Carbon	C
Carburetor	CARB
Carburize	CARB
Carriage	CRG
Carton	CTN
Case Harden	CH
Cast (Used with Other Materials	C
Cast Iron	CI
Cast Iron Pipe	CIP
Cast Steel	CS
Casting	CSTG
Castle Nut	CAS NUT
Catalogue	CAT.
Cathode Ray Tube	CRT
Cavity	CAV
Ceiling	CLG
Cement	CEM
Center	CTR
Center Line	CL
Center of Gravity	CG
Center Punch	CP
Center to Center	C to C
Centigrade	C
Centrifugal	CENT.
Ceramic	CER
Chamfer	CHAM
Change	CHG
Change Notice	CN
Change Order	CO
Channel	CHAN
Check	CHK
Chemical	CHEM
Chord	CHD
Chrome Molybdenum	CR MOLY
Chromium Plate	Cr PL
Chrome Vanadium	CR VAN
Circle	CIR
Circuit	CKT
Circular Pitch	CP
Circumference	CIRC
Clamp	CLP
Cleanout	CO
Clearance	CL
Clockwise	CW
Closet	CLO
Coated	CTD
Coaxial	COAX
Cold Drawn	CD
Cold Drawn Steel	CDS
Cold Finish	CF
Cold Punched	CP
Cold Rolled Steel	CRS
Column	COL
Combination	COMB.
Command	COM
Commercial	COML
Commercial Quality	CQ
Commutator	COMM
Companion	COMP
Compensator	COMP
Complete	COMPL
Compound	COMP
Compressor	COMPR
Concentric	CONC

Concrete	CONC
Condition	COND
Constant	CONST
Construction	CONSTR
Container	CNTR
Continue	CONT
Continuous Wave	CW
Contract	CONTR
Contractor	CONTR
Contractor Furnished Equipment	CFE
Control	CONT
Conveyor	CNVR
Copper Plate	COP PL
Correct	CORR
Corrosion Resistant	CRE
Corrosion Resistant Steel	CRES
Corrugate	CORR
Cotter	COT
Cotton Webbing	COT. WEB.
Counter Clockwise	CCW
Counter Electromotive Force	CEMF
Counterbore	CBORE
Counterdrill	CDRILL
Counterpunch	CPUNCH
Countersink	CSK
Countersink Other Side	CSKO
Coupling	CPLG
Cowling	COWL
Crankcase	CRKC
Cross Arm	XARM
Cross Section	XSECT
Cubic	CU
Cubic Feet per Minute	CFM
Cubic Feet per Second	CFS
Cubic Foot	CU FT
Cubic Inch	CU IN.
Cubic Inches per Minute	CIPM
Cubic Yard	CU YD
Current	CUR.
Cyanide	CYN
Cycle	CY
Cylinder	CYL

D

Decalcomania	DECAL
Decibel	DB
Decimal	DEC
Deck	DK
Dedendum	DED
Deep Drawn	DD
Degree	(°) DEG
Density	D
Department	DEPT
Design	DSGN
Designed Water Line	DWL
Detail	DET
Detent	DET
Develop	DEV
Diagonal	DIAG
Diagram	DIAG
Diameter	DIA
Diametal Pitch	DP
Diaphragm	DIAPH
Direct Current	DC
Discharge	DISCH
Ditto	DO.
Division	DIV
Dovetail	DVTL
Dowel	DWL
Downspout	DS
Drafting	DFTG
Draftsman	DFTSMN
Drain	DR
Drawing	DWG
Drawing List	DL
Drill	DR
Drill Rod	DR
Drive	DR
Drive Fit	DF
Drop Forge	DF
Duplex	DX
Duplicate	DUP

E

Each	EA
Eccentric	ECC
Effective	EFF
Effective Horsepower	EHP
Efficiency	EFF
Elastic Limit	EL
Elbow	ELL.
Electric	ELEC
Electromotive Force	EMF
Elementary	ELEM
Elevate	ELEV
Elevation	EL
Elongation	ELONG
Enamel	E
Enclose	ENCL
End to End	E to E
Engine	ENG
Engineer	ENGR
Engineering	ENGRG
Engineering Change Order	ECO
Entrance	ENT
Envelope	ENV
Equal	EQ
Equipment	EQUIP.
Equipment & Spare Parts	EASP
Equivalent	EQUIV
Estimate	EST
Excavate	EXC
Exchange	EXCH
Executive	EXEC
Exhaust	EXH
Existing	EXIST.
Expansion (Joint)	EXP
Explosion Proof	EP
Extension	EXT
Exterior	EXT
Extra Heavy	X HVY
Extra Strong	X STR
Extrude	EXTR

F

Fabricate	FAB
Face to Face	F to F
Fanrenheit	F
Far Side	FS
Farads	F
Federal	FED.
Federal Specification	FS
Federal Stock Number	FSN
Feeder	FDR
Feet	(') FT
Feet Board Measure	FBM
Feet per Minute	FPM
Feet per Second	FPS
Female	FEM
Fiber	FBR
Figure	FIG.
Fillet	FIL
Fillister	FIL
Filter	FLT
Finish	FIN.
Finish All Over	FAO
Fire Door	F DR
Fire Hose	FH
Fire Hydrant	FHY
Fireproof	FPRF
Fitting	FTG
Fixture	FIX.
Flange	FLG
Flashing	FL
Flat	F
Flat Head	FH
Flexible	FLEX.
Floor	FL
Floor Drain	FD
Floor Line	FL
Flow Meter	FM
Fluid	FL
Fluorescent	FLUOR
Foot	(') FT
Foot Candle	FC
Forged Steel	FST
Forging	FORG
Forward	FWD
Foundation	FDN
Foundry	FDRY
Fractional	FRAC
Fractional Horsepower	FHP
Frame	FR
Framework	FRWK
Frequency	FREQ
Frequency Modulation	FM
Front	FR
Furnish	FURN
Fuselage	FUS

G

Gage	GA
Gallon	GAL
Gallons per Minute	GPM
Galvanize	GALV
Galvanized Iron	GI
Galvanized Steel	GS
Galvanized Steel Wire Rope	GSWR
Gasket	GSKT
General	GEN
Generator	GEN
Girder	G
Glass	GL
Government	GOVT
Grade	GR
Grade Line	GL
Graphic	GRAPH.
Graphite	GPH
Grid	G
Grind	GRD
Grommet	GROM
Groove	GRV
Ground	GND or GRD

H

Half-Hard	½H
Half-Round	½RD
Handle	HDL
Hanger	HGR
Hard	H
Hard-Drawn	HD
Harden	HDN
Hardware	HDW
Head	HD
Headless	HDLS

Heat	HT
Heat Exchanger	HE
Heat Treat	HT TR
Heavy	HVY
Height	HGT
Henries	H
Hexagon	HEX.
Hexagon	HEX
High Frequency	HF
High-Pressure	HP
High-Speed	HS
High-Speed Steel	HSS
High-Tensile Cast Iron	HTCI
High-Tensile Steel	HTS
Highway	HWY
Horizontal	HORIZ
Horsepower	HP
Hot Rolled	HR
Hot Rolled Steel	HRS
Hot Water Tank	HWT
Hour	HR
Hundredweight	CWT
Hydraulic	HYD
Hydrostatic	HYDRO

I

Ignition	IGN
Impregnate	IMPG
Inboard	INBD
Incandescent	INCAND
Inch	(") IN.
Inches per Minute	IPM
Incorporated	INC
Indicate	IND
Inductance	L
Industrial	IND
Inside Diameter	ID
Inspect	INSP
Install	INSTL
Insulate	INS
Intensifier	INT
Interchangeable	INTCHG
Intercommunication	INTERCOM
Interior	INT
Intermediate Frequency	IF
Internal	INT
International Annealed Copper Standard	IACS
International Pipe Standard	IPS
Interphone Control Station	ICS
Intersect	INT
Iron	I
Iron-Pipe Size	IPS
Irregular	IRREG
Issue	ISS

J

Job Order	JO
Joint	JT
Journal	JNL
Junction	JCT
Junior	JR

K

Key	K
Keyseat	KST
Keyway	KWY
Kiln-Dried	KD
Kilocycle	KC
Kilohm	K
Kilowatt Hour	KWH
Kilovolt	KV
Kip (1000 lb)	K
Knocked Down	KD
Knockout	KO

L

Laboratory	LAB
Lacquer	LAQ
Laminate	LAM
Lateral	LAT.
Lavatory	LAV
Lead Covered	LC
Leading Edge	LE
Left	L
Left Hand	LH
Length	LG
Length Over All	LOA
Letter	LTR
Light	LT
Limit	LIM
Line	L
Linear	LIN
Liquid	LIQ
Lithograph	LITHO
Locate	LOC
Long	LG
Loudspeaker	LS
Low Frequency	LF
Low Pressure	LP
Low-Speed	LS
Lubricate	LUB
Lubricator	LUB
Lumber	LBR

M

Machine	MACH.
Machine Steel	MS
Magnaflux	M
Maintenance	MAINT
Male & Female	M&F
Malleable	MALL.
Malleable Iron	MI
Manual	MAN.
Manufacture	MFR
Manufactured	MFD
Manufacturing	MFG
Marine	MAR.
Mark	MK
Material	MATL
Material List	ML
Maximum	MAX
Mechanical	MECH
Median	MED
Megacycle	MC
Megohm	MEG
Memorandum	MEMO
Metal	MET.
Meter (Instrument or Measure of Length	M
Metered Flow	MF
Mezzanine	MEZZ
Micro (10^{-6})	μ or U
Micrometer	MIC
Mile	MI
Miles per Gallon	MPG
Miles per Hour	MPH
Military	MIL
Millihenries	MH
Millimeter	MM
Minimum	MIN
Minute	(') MIN
Miscellaneous	MISC
Mixture	MIX.
Model	MOD
Mold Line	ML
Molded	MLD
Molding	MLDG
Month	MO
Monument	MON
Morse Taper	MOR T
Motor	MOT
Mounted	MTD
Mounting	MTG
Multiple	MULT
Music Wire Gage	MWG

N

Name Plate	NP
National	NATL
National Aircraft Standards	NAS
National Electrical Code	NEC
Natural	NAT
Naval Architect	NA
Near Face	NF
Near Side	NS
Negative	NEG
New British Standard (Imperial Wire Gage)	NBS
Nipple	NIP.
Nominal	NOM
Noon	M
Normal	NORM.
North	N
Not to Scale	NTS
Number	NO.

O

Obsolete	OBS
Octagon	OCT
Office	OFF.
Ohms	Ω
On Center	OC
One Pole	1 P
Opening	OPNG
Opposite	OPP
Optical	OPT
Ordnance	ORD
Original	ORIG
Oscillator	OSC
Ounce	OZ
Outboard	OUTBD
Outlet	OUT.
Outside Diameter	OD
Overall	OA
Overflow	OVFL
Overhead	OVHD
Oxidized	OXD

P

Pack	PK
Packing	PKG
Page	P
Pair	PR
Panel	PNL
Paragraph	PARA
Parallel	PAR.
Parkway	PKWY
Part	PT
Passage	PASS.
Patent	PAT.
Pattern	PATT
Pack	PK
Penny (Nails, etc)	d
Permanent	PERM
Perpendicular	PERP
Phenolic	PHEN
Phosphor Bronze	PH BRZ
Pick Up	PU

Piece	PC
Piece Mark	PC MK
Pint	PT
Pipe Tap	PT
Pitch	P
Pitch Circle	PC
Pitch Diameter	PD
Plaster	PLAS
Plastic	PLSTC
Plate	PL
Plumbing	PLMB
Pneumatic	PNEU
Point	PT
Pole	P
Polish	POL
Porcelain	PORC
Port	P
Portable	PORT
Position	POS
Potential	POT.
Potentiometer	POT
Pound	LB
Pounds per Square Inch	PSI
Power	PWR
Power Factor	PF
Precast	PRCST
Prefabricated	PREFAB
Preferred	PFD
Prepare	PREP
Press	PRS
Pressure	PRESS.
Pressure Angle	PA.
Pressure Controlled	PC
Pressure Switch	PS
Primary	PRIM.
Process	PROC
Production	PROD.
Production Order	PO
Profile	PF
Project	PROJ
Proposed	PROP.
Publication	PUB.
Pull Box	PB
Pull Switch	PS
Pulse Modulation	PM
Pump. Fixed Displacement	PF
Pump. Variable Displacement	PV
Punch	PCH
Purchase	PUR
Push Button	PB

Q

Quadrant	QUAD.
Quality	QUAL
Quantity	QTY
Quart	QT
Quarter-Hard	¼ H
Quarter-Round	¼ RD
Quartermaster Corps	QMC

R

Radial	RAD
Radio Frequency	RF
Radius	R
Railroad	RR
Railway	RY
Ream	RM
Reassemble	REASM
Received	RECD
Receptacle	RECP
Record	REC
Rectangle	RECT
Rectifier	RECT
Reduce	RED.
Reducer	RED.
Reference	REF
Reference Line	REF L
Register	REG
Regulator	REG
Reinforce	REINF
Relative Humidity	RH
Relay	REL
Relief	REL
Remove	REM
Renewable	REN
Reproduce	REPRO
Require	REQ
Required	REQD
Requisition	REQ
Resistance	R
Retard	RET.
Return	RET.
Reverse	REV
Revise	REV
Revolution	REV
Revolutions per Minute	RPM
Right	R
Right Hand	RH
Rivet	RIV
Rockwell Hardness	RH
Roller Bearing	RB
Root Diameter	RD
Root Mean Square	RMS
Round	RD

S

Schedule	SCH
Schematic	SCHEM
Screw	SCR
Sea Level	SL
Seamless	SMLS
Secondary	SEC
Section	SECT
Semi-Finished	SF
Semi-Fixed	SFXD
Semi-Steel	SS
Separator	SEP
Serial	SER
Serrate	SERR
Servo	SERV
Set Screw	SS
Sewer	SEW.
Shaft	SFT
Sheathing	SHTHG
Sheet	SH
Shop Order	SO
Short Wave	SW
Shoulder	SHLD
Shut Off Valve	SOV
Side	S
Similar	SIM
Sink	SK
Sketch	SK
Sleeve	SLV
Sleeve Bearing	SB
Slotted	SLOT.
Small	SM
Socket	SOC
Soft	S
Soil Pipe	SP
Solenoid	SOL
Solenoid Controlled, Pilot Operated	SOL PLT
Space	SP
Spare	SP
Special	SPL
Special Treatment Steel	STS
Specific Gravity	SP GR
Specification	SPEC
Speed	SP
Spherical	SPHER
Split Phase	SP PH
Spot Faced	SF
Spring	SPG
Square	SQ
Squirrel Cage	SQ CG
Stainless	STN
Stainless Steel	SST
Stairway	STWY
Standard	STD
Starboard	STBD
Station	STA
Steel	STL
Stock	STK
Storage	STOR
Straight	STR
Strainer	STR
Street	ST
Stress Anneal	SA
Strip	STR
Structural	STR
Substitute	SUB
Substructure	SUBSTR
Summary	SUM
Supercharge	S CHG
Superintendent	SUPT
Supersede	SUPSD
Superstructure	SUPRSTR
Supervise	SUPV.
Supplement	SUPP
Supply	SUP
Surface	SURF
Survey	SURV
Switch	SW
Switch and Relay Types	
Single Pole Switch	SP SW
Single Pole Single Throw Switch	SPST SW
Single Pole Double Throw Switch	SPDT SW
Double Pole Switch	DP SW
Double Pole Single Throw Switch	DPST SW
Double Pole Double Throw Switch	DPDT SW
Triple Pole Switch	3P SW
Triple Pole Single Throw Switch	3PST SW
Triple Pole Double Throw Switch	3PDT SW
4 Pole Switch	4P SW
4 Pole Single Throw Switch	4PST SW
4 Pole Double Throw Switch	4PDT SW
etc	
Switchboard	SWBD
Switchgear	SWGR
Symbol	SYM
Symmetrical	SYM
Synchronous	SYN
Synthetic	SYN
System	SYS

T

Tabulate	TAB
Tachometer	TACH
Tangent	TAN.
Taper	TPR
Technical	TECH
Technical Manual	TM
Technical Report	TR
Tee	T

Teeth ..... T
Teeth per Inch ..... TPI
Temperature ..... TEMP
Template ..... TEMP
Tensile Strength ..... TS
Tension ..... TENS.
Terminal ..... TERM.
That is ..... i.e.
Thermal ..... THRM
Thick ..... THK
Thousand ..... M
Thousand Foot Pounds .. KIP-FT
Thousand Pound ..... KIP
Thread ..... THD
Threads per Inch ..... TPI
Through ..... THRU
Timber ..... TMBR
Time ..... T
Time Delay ..... TD
Tobin-Bronze ..... TOB BRZ
Toggle ..... TGL
Tolerance ..... TOL
Tongue & Groove ..... T&G
Tool Steel ..... TS
Tooth ..... T
Total ..... TOT.
Total Indicator Reading .... TIR
Trailing Edge ..... TE
Training ..... TNG
Transfer ..... TRANS
Transformer ..... TRANS
Transistor ..... Q
Transmission ..... XMSN
Transportation ..... TRANS
Truss ..... T
Tubing ..... TUB.
Typical ..... TYP

U

Ultimate ..... ULT
Ultra-High Frequency .... UHF
United States Gage ..... USG
United States Standard .... USS
Universal ..... UNIV

V

Vacuum ..... VAC
Vacuum Tube ..... VT
Vacuum Tube Voltmeter ..... VTVM
Valve V
Vandyke ..... VD
Vapor Proof ..... VAP PRF
Varnish ..... VARN
Velocity V
Vent Pipe ..... VP
Ventilate ..... VENT.
Vertical ..... VERT
Very-High Frequency ..... VHF
Vibrate ..... VIB
Video Frequency ..... VF
Vitreous ..... VIT
Volt V
Voltmeter ..... VM
Volume ..... VOL

W

Wall ..... W
Washer ..... WASH.
Water Heater ..... WH
Water Line ..... WL
Watt ..... W
Week ..... WK
Weight ..... WT
West ..... W
Wheel Base ..... WB
Width ..... W
Wire ..... W
With ..... W/
Without ..... W/O
Woodruff ..... WDF
Wrought ..... WRT
Wrought Iron ..... WI

X Y Z

Yard ..... YD
Year ..... YR

ABBREVIATIONS RELATING TO SCREW THREADS

Screw-Thread Standards
American National Coarse Thread ..... NC
American National Fine Thread ..... NF
American National Extra Fine Thread ..... NEF
American National Special Pitch, etc. Thread ..... NS
American National Pitch Thread . 8N, 12N, or 16N
American National Acme Thread ..... NA
American National Taper Pipe Thread ..... NPT
American National Straight Pipe Thread in Pipe Couplings ..... NPSC
American National Straight Pipe Threads for Dry Seal Pressure Tight Joint . NPSF
American National Taper Pipe Threads for Dry Seal Pressure Tight Joints NPTF
American National Straight Pipe Threads for Mechanical Joints .. NPSM
American National Straight Pipe Threads for Locknuts and Locknut Pipe Threads ..... NPSL
American National Straight Pipe Threads for Hose Couplings and Nipples NPSH
American National Taper Pipe Threads for Railing Fixtures ..... NPTR
American Truncated Whitworth Coarse Thread ..... TWC
American Truncated Whitworth Fine Thread ..... TWF
American Truncated Whitworth Special Thread ..... TWS
Unified National Course Thread ..... UNC
Unified National Fine Thread ..... UNF
Unified National Special Thread ..... UNS

ENGINEERING SOCIETIES, TRADE ASSOCIATIONS, AND INSPECTION BUREAUS, ETC.

American Association of Engineers ..... AAE
American Boiler Manufacturers' Association & Affiliated Industries ... ABMA
American Bureau of Shipping ..... ABS
Air Conditioning & Refrigerating Machinery Association ..... ACRMA
American Chemical Society ..... ACS
American Concrete Institute ..... ACI
American Electrochemical Society ..... AES
American Electroplaters Society ..... AES
American Engineering Council ..... AEC
American Foundrymen's Association ..... AFA
American Gas Association ..... AGA
American Gear Manufacturers' Association .. AGMA
American Institute of Architects ..... AIA
American Institute of Chemical Engineers ... AICE
American Institute of Electrical Engineers .. AIEE
American Institute of Mining & Metallurgical Engineers ..... AIMME
American Institute of Steel Construction .... AISC
American Iron & Steel Institute ..... AISI
American Manufacturers Association ..... AMA

American Petroleum Institute ..... API
American Railway Engineering Association .. AREA
American Railway Bridge & Building Association . ARBBA
American Society of Aeronautical Engineers ..... ASAE
American Society of Civil Engineers ..... ASCE
American Society of Engineers and Architects ..... ASEA
American Society of Heating & Ventilating Engineers ..... ASHVE
American Society of Lubricating Engineers . ASLE
American Society of Mechanical Engineers .... ASME
American Society for Metals ..... ASM
American Society of Refrigerating Engineers .. ASRE
American Society of Safety Engineers ..... ASSE
American Society of Sanitary Engineering ..... ASSE
American Society for Steel Treating..... ASST
American Society for Testing and Materials . ASTM
American National Standards Institute ..... ANSI
American Standards Association ..... ASA
American Steel Foundrymen's Association ..... ASFA
American Telephone & Telegraph Co. ..... AT&TCO
American Transit Association ..... ATA
American Water Works Association ..... AWWA
American Welding Society . AWS
American Wood Preservers Association ... AWPA
Anti-Friction Bearing Manufacturers' Association ..... AFBMA
Association of American Railroads ..... AAR
Association of American Steel Manufacturers ... AASM
Association of Iron & Steel Engineers ..... AISE
Automobile Manufacturers' Association ..... AMA
Canadian Standards Association ..... CSA
Canadian Lumbermen's Association ..... CLA
Compressed Air Institute .. CAI
Department of Defense ... DOD
Edison Electric Institute .... EEI
Electrochemical Society ..... ES
Gas Appliances Manufacturers' Association.. GAMA
Hydraulic Institute ..... HI
Illuminating Engineering Society ..... IES
Institute of Radio Engineers ..... IRE
Institute of Traffic Engineers ..... ITE
Insulated Power Cable Engineers' Association .. IPCEA
International Standardization Organization ..... ISO
Joint Electron Tube Engineering Council .. JETEC
Joint Industrial Council ..... JIC
Manufacturers Standardization Society of the Valve and Fittings Industry ..... MSS
Military Standards (DOD) . MIL
Mill Mutual Fire Prevention Bureau .. MMFPB
National Advisory Committee for Aeronautics ..... NACA
National Aeronautics and Space Administration .. NASA
National Aircraft Standards ..... NAS
National Bureau of Standards ..... NBS
National Association of Manufacturers ..... NAM
National Conservation Bureau ..... NCB
National Electrical Manufacturers Association.. NEMA
National Hardwood Lumber Association ... NHLA
National Housing Agency.. NHA
National Lumber Manufacturers Association .... NLMA
National Machine Tool Builders' Association. NMTBA
National Petroleum Association ..... NPA
National Safety Council ... NSC
Oil Heat Institute of America ..... OHIA
Radio Manufacturers' Association ..... RMA
Refrigeration Equipment Manufacturers' Association ..... REMA
Society for the Advancement of Management ..... SAM
Society of Automotive Engineers ..... SAE
Society of Fire Engineers... SFE
Society of Industrial Engineers ..... SIE
Society of Military Engineers ..... SME
Society of Manufacturing Engineers ..... SMI
Society of Tractor Engineers ..... STL
Standards Engineers Society ..... SES
Underwriters' Laboratories, Inc. ..... UL